ROHINI COLLEGE OF ENGINEERING AND TECHNOLOGY

Kanyakumari Main Road, Near Anjugramam, Palkulam, Tamil Nadu 629401

Department of Mechanical Engineering

VALUE ADDED COURSE ON CREO PARAMERTICS

SYLLABUS

CHAPTER 1
CREO PARAMETRIC INTERFACE

Introducing CREO 2.0

CREO PARAMETRIC INTERFACE

The main interface includes the following areas :-

- GRAPHICS WINDOW
  The working area of creo parametric in which we create and modify Creo Parametric models such as parts , assemblies and drawings .

- QUICK ACCESS TOOLBAR
  By default , the quick access toolbar is located at the top of interface . It contains a commonly used set of commands that are independent of the tab currently displayed in the ribbon .

- RIBBON
  A context sensitive menu across the top of the interface that contains the majority of commands you use in Creo Parametric . The ribbon arranges commands into logical tasks through tabs and groups .

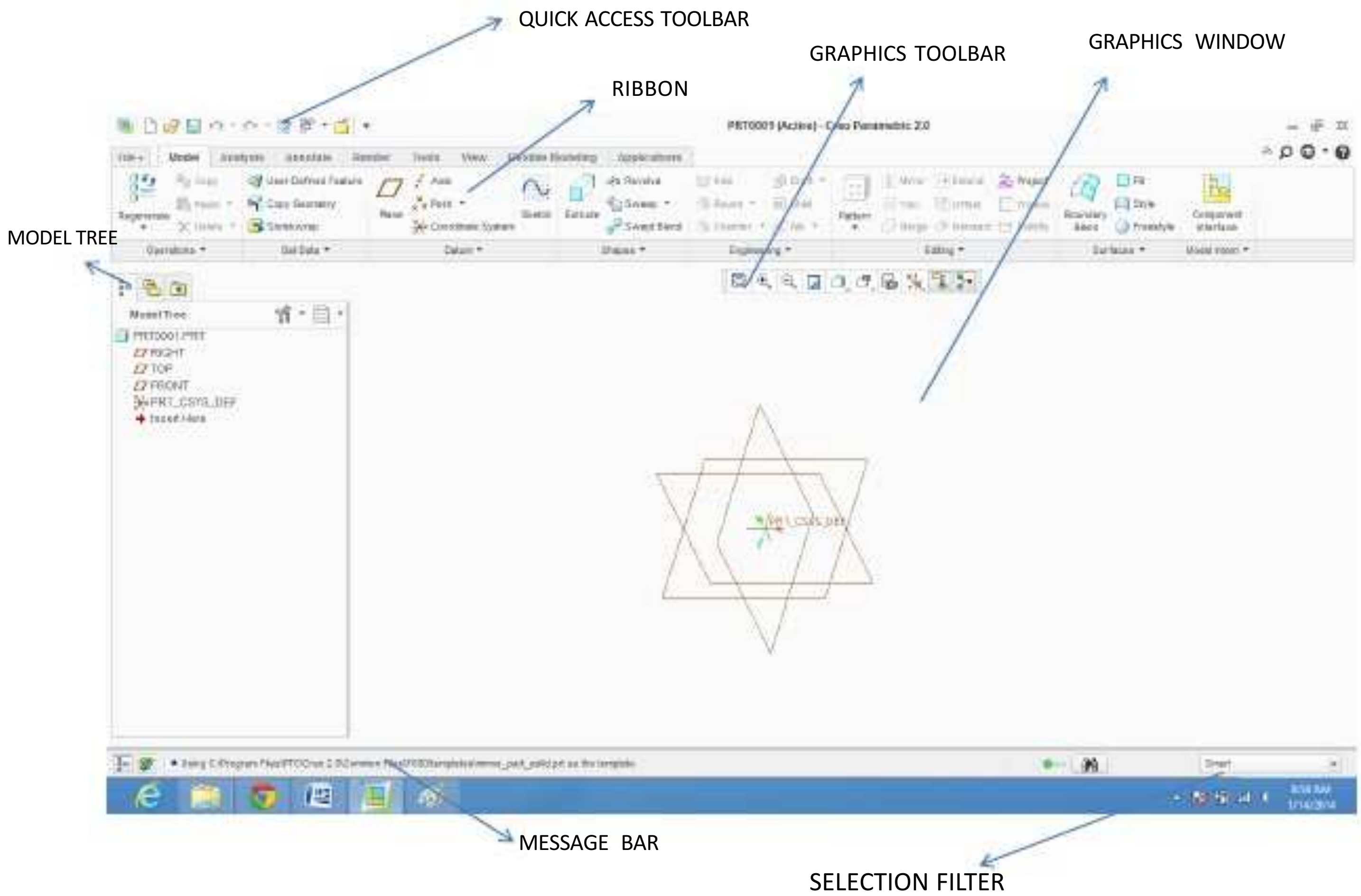
- DASHBOARD

The dash board provides you with controls , inputs , status and guidance for performing a task such as creating or editing a feature .

- DIALOGUE BOX
  Content sensitive windows which display and prompt you for additional information

- STATUS BAR
  Located at the bottom of the interface, the status bar contains icons for toggling the model tree and web browser planes on and off. It also contains the message log , regeneration manager , 3D box selector and selection filter

- MESSAGE LOG
  The message log provides you with prompts , feedback , and message from Creo Parametric


QUICK ACCESS TOOLBAR
GRAPHICS TOOLBAR
GRAPHICS WINDOW
RIBBON
MODEL TREE
MESSAGE BAR
SELECTION FILTER

CHAPTER 2
CREO PARAMETRIC CONCEPTS, MANAGING FILES IN CREO

Creo Parametric enables you to create solid model representations of our part and assembly models . These virtually design models can be used to easily visualize and evaluate our design before costly prototypes are manufactured .

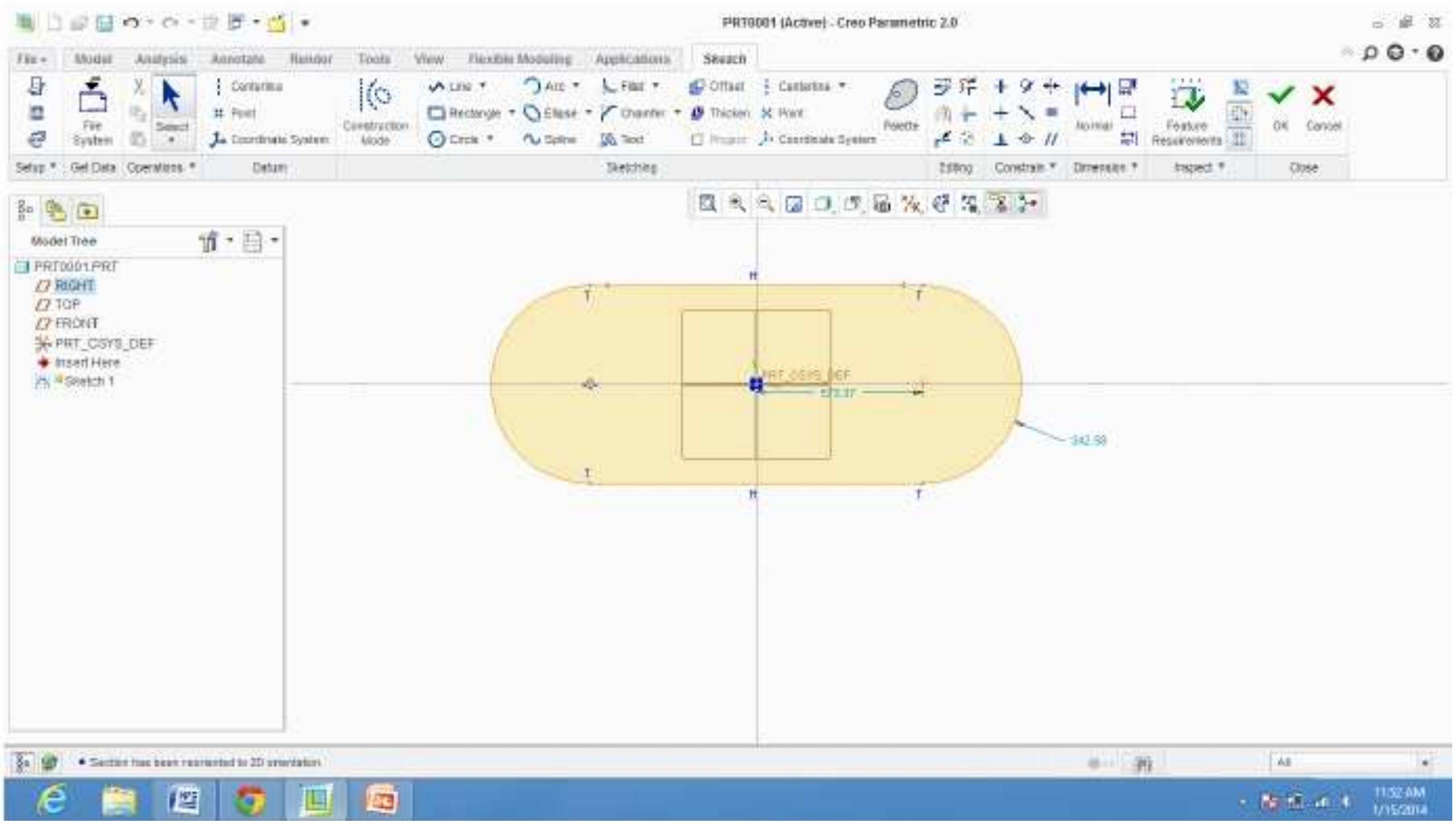
The model contains material properties such as mass , volume , center of gravity and surface area . As features are added or removed from the model , these properties update . For example if we add a hole to a model its mass decreases .

DESIGN INTENT:- Designs are created for a purpose. Design intent is the intellectual arrangements of features and dimensions of a design. Design intent governs that relationship between features in a part and parts in assemblies. The intent of each component of a design is to work as a solution to the design problem.

CONCEPT OF CREO :-

- Creo is a sketch based, feature based Parametric ,3D modeling software. Which is developed by the Parametric Technologies Corporation(PTC).Creois having the Quality of Parent-child relationship and Bi-directional Associativity.

<u>SKETCH</u>:- It is a Combination of 2D elements which is required to create the 3D models.



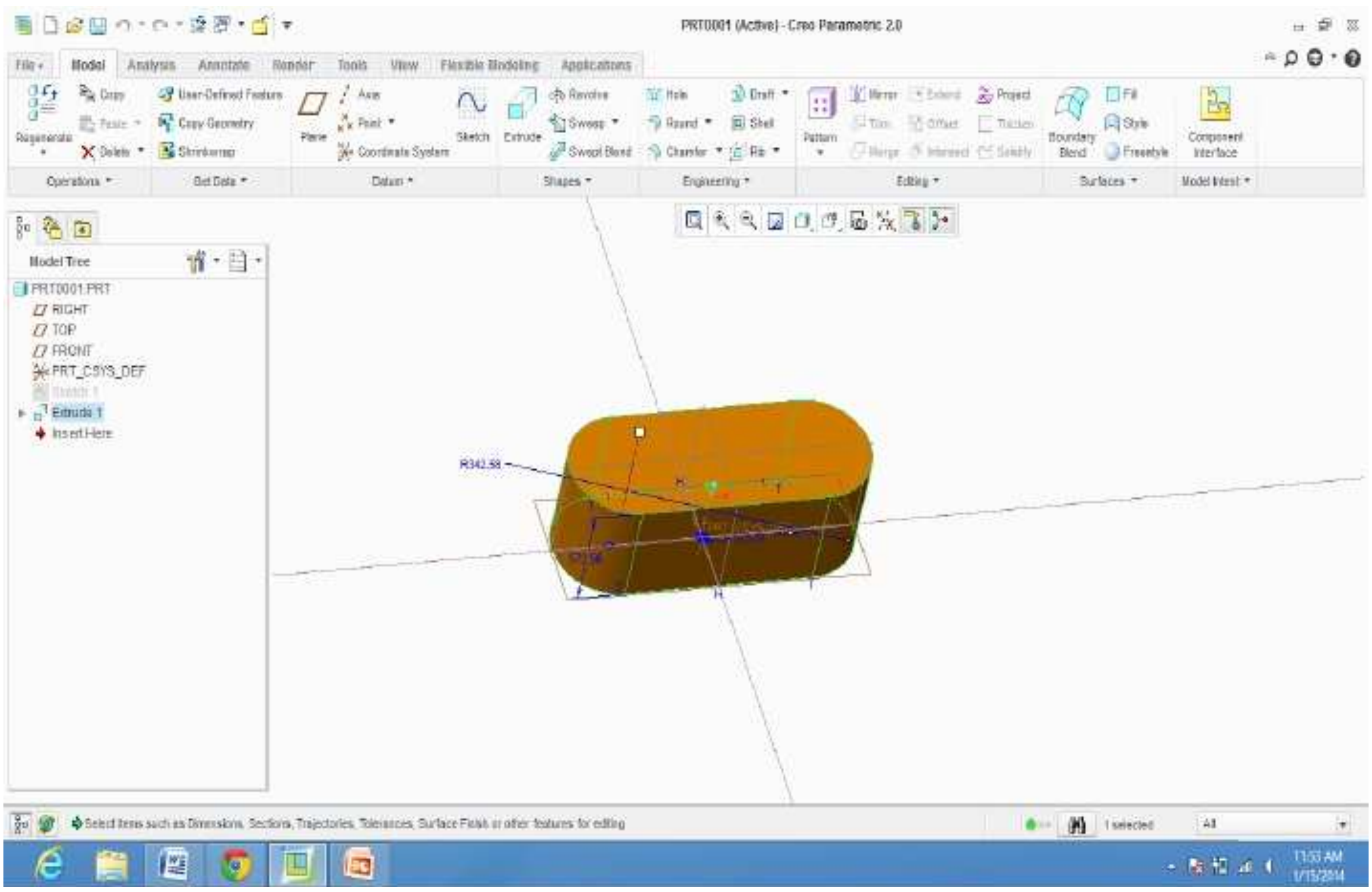
<u>FEATURE</u>:- It is a basic building block of a model . Features combine together to form a model

<u>FEATURE BASED MEANING</u>:

- The models are constructed using a series of easy to understand features rather than confusing mathematical shapes and entities. Individually each feature is simple , however as they are added together they form complex parts and assemblies .

PARAMETRIC :-

- We can change the shape and size of the Geometry whenever we required, without considering its original shape and size.
- Parametric nature enables you to easily capture design intent and make design changes .



BI-DIRECTIONAL ASSOCIATIVITY :-

- Related to other module if we make any change in any of the module of the cad s/w , same change will reflect to us in other modules also this behaviour of the s/w is called Bi-directional Associativity.

PARENT CHILD RELATIONSHIP BETWEEN PARTS AND ITS FEATURES

- Feat -1
- Feat-2
- Feat-3
- Feat-4
- Feat-5
- Feat-1 is the Parent to the child Feat-2 ,Feat-2 is the parent to Feat-3 and so on
- Parent can stay without child but child cannot stay without Parent.

BETWEEN ASSEMBLIES AND ITS PARTS

- Part -1
- Part-2
- Part-3
- Part-4
- Part-5
- Part-1 is the Parent to the child Feat-2 ,Part-2 is the parent to Feat-3 and so on
- Parents can stay without child but child cannot stay without Parent.

MANAGING FILES IN CREO

Pro-e is a memory-based system , meaning that files are stored within RAM while you work on them.

There are 2 methods to erase models from session

1: File > manage session >Erase Current : Only the model in the current window is erased from system memory .

2: File >manage session > Erase Not Displayed : Only erases from system memory those models that are not found in any pro/e windows.

NOTE: Erasing models does not delete them from the hard drive , it only removes from session .

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CHAPTER 3
CREATING AND MODIFYING SKETCHES

A sketch is a combination of 2d entities that is required to create 3d models .

2-d sketches are :

1> Sketched on a 2-D plane
2> Placed on a 3-D model
3> Used to create solid features

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2-d sketches are :

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2> Placed on a 3-D model
3> Used to create solid features

How to invoke sketcher ?

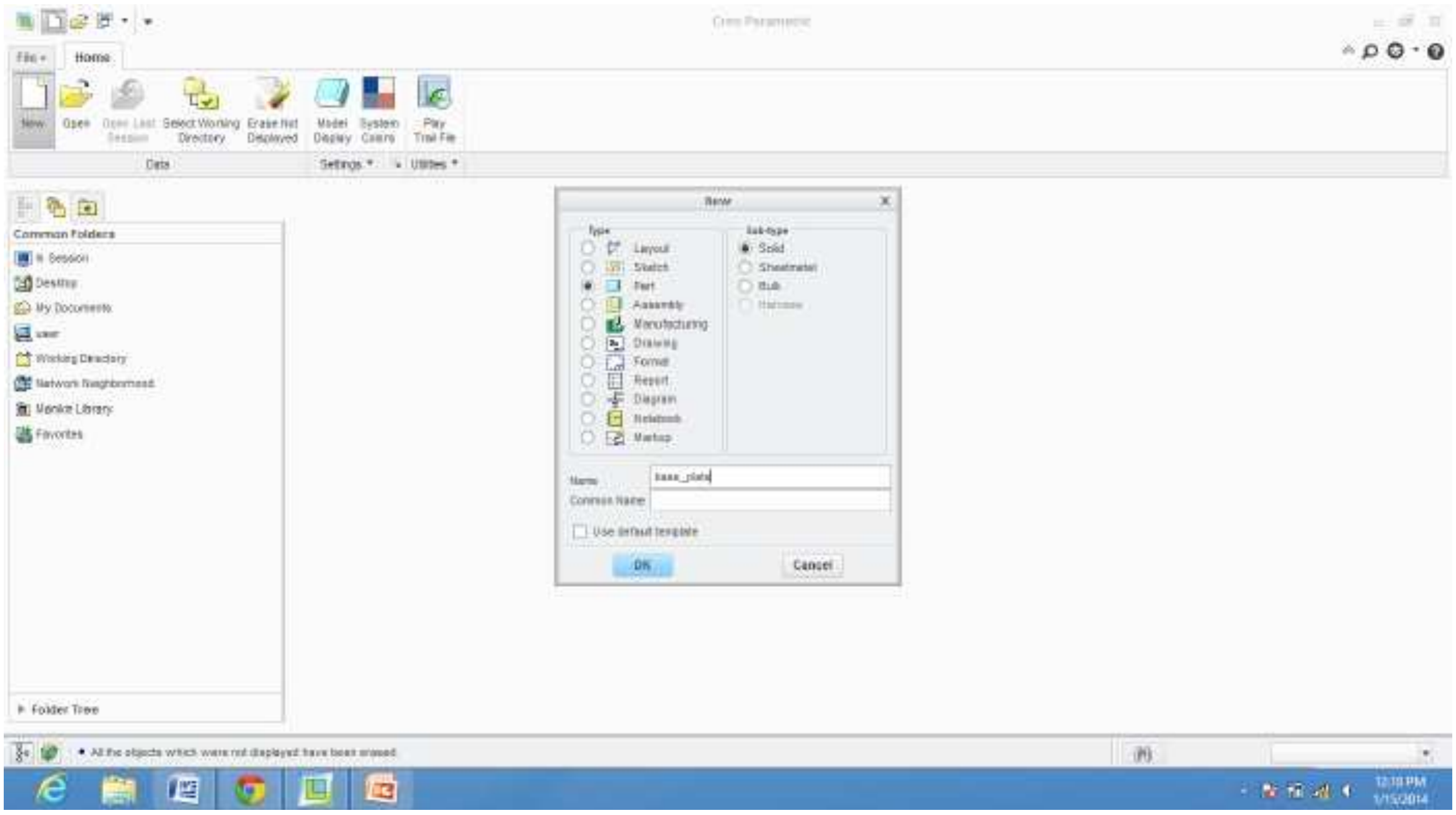
Step 1: Specify working directory . Here working directory serves as a destination for opening and closing of files .

Go to working directory from ribbon > select destination (may be any of drives i.e. c:,d:,e)

Step 2: Rt click > new folder > name the folder (for example : akash)

Step 3: Double click on folder

Step 4: Goto new > part (type): solid (sub type) , uncheck the ‘ use default templelate ‘ > name the file> click ok> mmns part solid



Step 5:click on sketch view to make the plane parallel to screen

Step 6: Sketch> select sketching plane > sketch view

Center line – center line is used to enforce symmetry in sketches . It is highly preferable to draw centerline in sketches .

Mouse clicks :

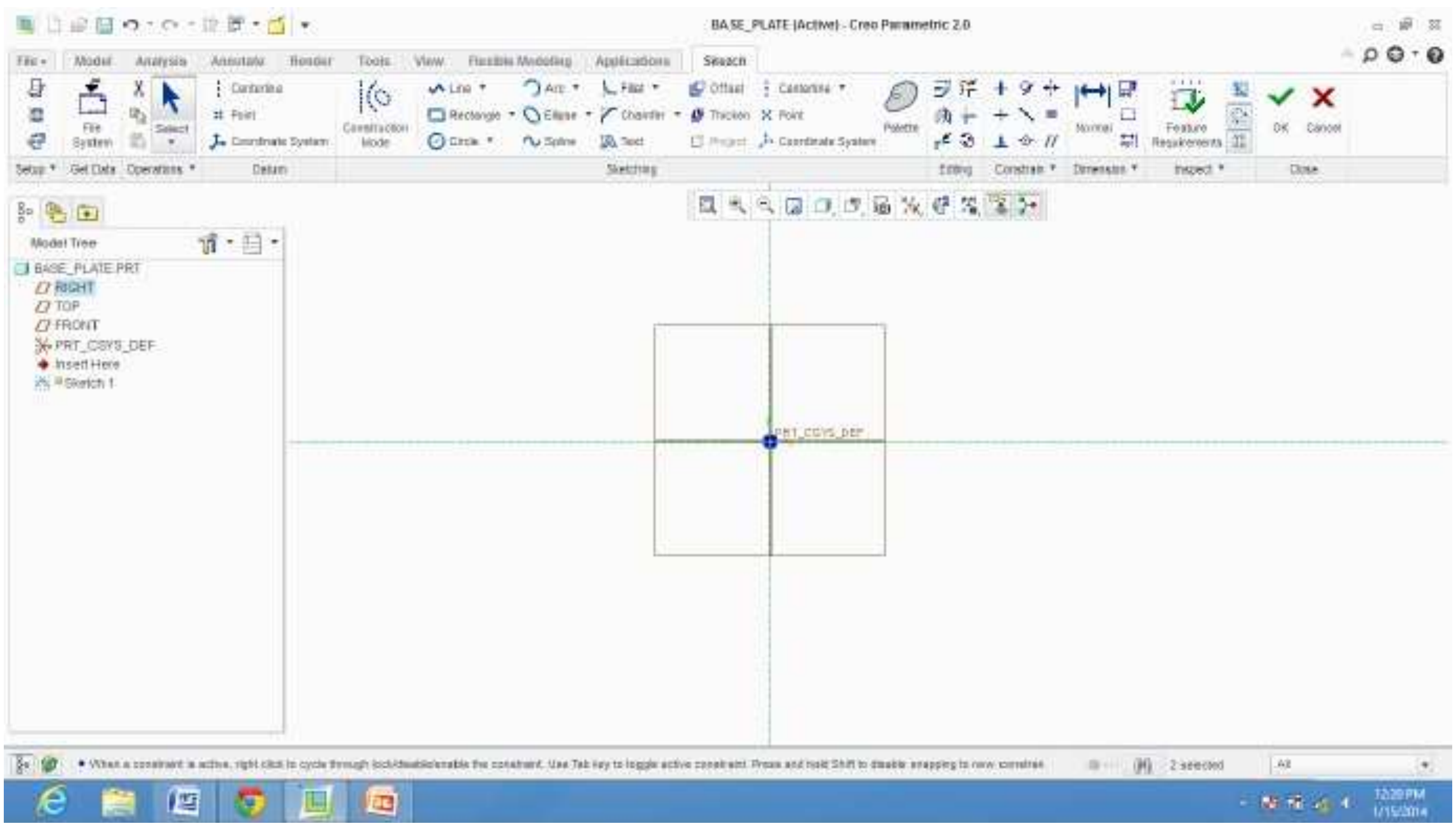
Mid button scroll:- zoom in/out

Shift+mid button:- pan

Continue mid button:- orient

Sketcher tools for creating 2d entities :

1> Line
2> Rectangle
3> Circle
4> Arc
5> Ellipse
6> Spline



Click on tool > left click > left click > mid button to exit

After creating sketch > click done on ribbon> click save> ok> click close > erase not displayed >ok

How design intent is captured in sketches ?

1> <u>By applying dimensional constraints</u>

By default all dimensions are weak and denoted by light blue in color. Our work is to make weak dimensions strong which is denoted by dark blue in colour .This can be done by left clicking on weak dimensions and assigning value then pressing enter . Software is having intent manager which tries to capture your design intent . If dimensions are not provided by intent manager then click on normal on the ribbon and provide dimensions

Left click on normal >left click on entity > mid button to display dimensions

For angular dimensions
Left click on first entity > left click on second entity > mid button

2> By applying geometrical constraints

Types of geometrical constraints

Vertical
Horizontal
Perpendicular
Tangent
Mid point
Coincident
Symmetric
Equal
Parallel

If dimensions are weak on two lines then after making equal the line dimensions will be mean of both .

If dimensions is strong then other line will be equal with reference to strong line

If we have already applied V & H in lines then after applying perpendicular , it will be a case of over constrained .

Note : All dimensions in sketch should be fully constrained which means geometrical as well as dimensional .

Editing tools are delete segment , mirror , rotate resize , divide , modify

INSPECTION : To ensure sketch is closed and doesn't form intersecting loops software provides inspection tools . These are :-

1> HIGHLIGHT OPEN ENDS :- If ends of loop are open it gets highlighted with red shade.
2> SHADE CLOSED LOOPS :- Shades the loop only when it is closed
3> OVELAPPING GEOMETRY :- Displays overlapping geometry with red colour.

CHAPTER 4
CREATING EXTRUDES AND CHAMFERS

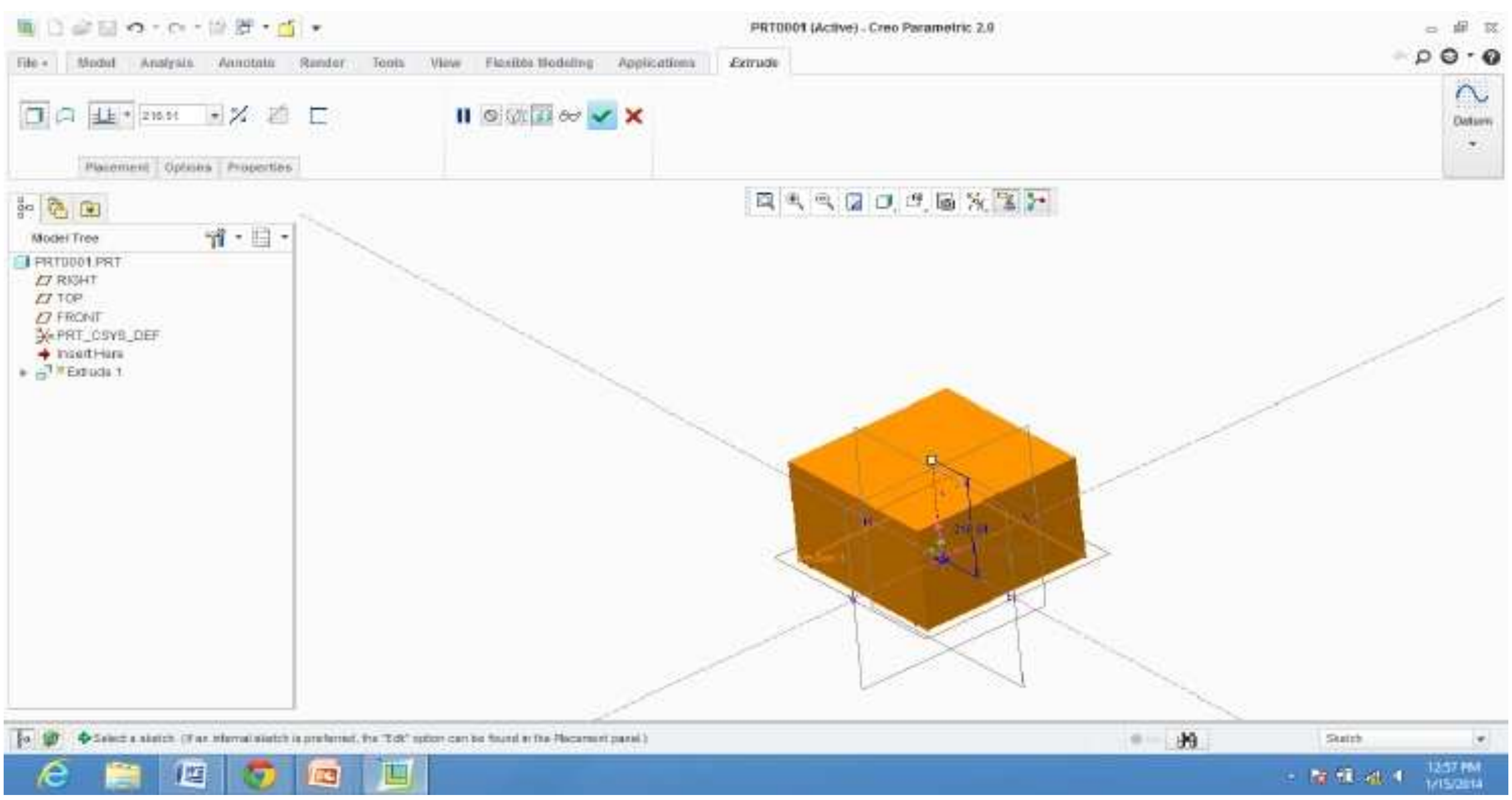
It's an addition or removal of material in any of the direction which is Normal to the sketching plane.

You need to create a <u>sketch that</u> <u>should be closed</u>.

If geometry is open it will extrude as a surface and if it is closed it will extrude as solid.

Steps involved:

Got to extrude tool > Placement > define> select plane to sketch



EXTRUDE DASHBOARD OPTIONS

Blind depth

Symmetric

To next

Thru until

To surface

To plane

REFERENCES

To create solid models we may require several references such as edge, point , plane , curve . The reference icon on ribbon helps you take references .

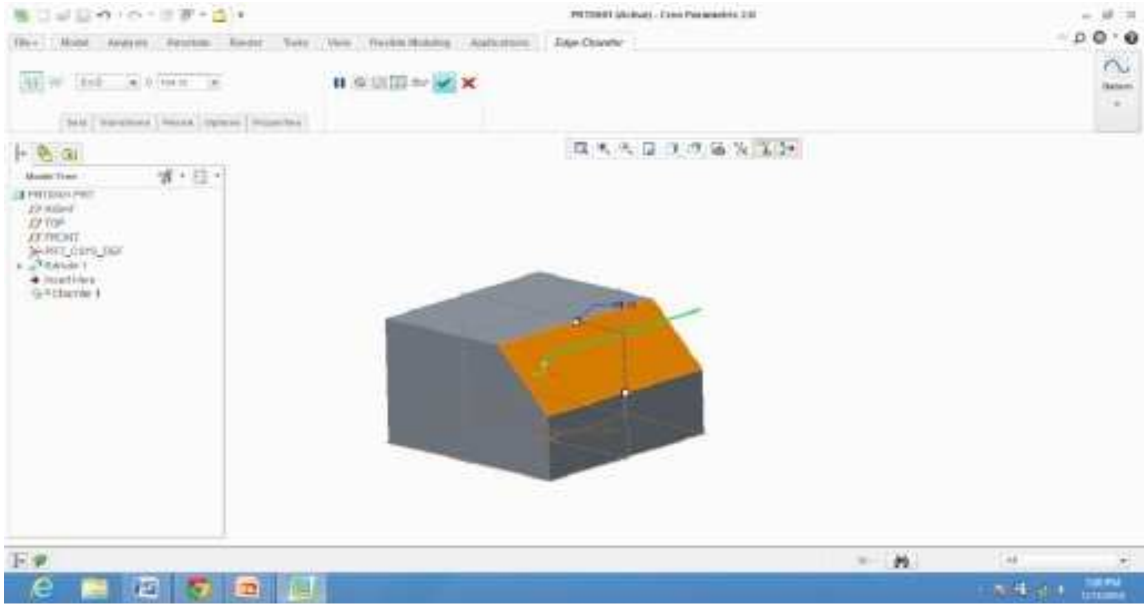
SELECTION FILTER

As the name suggests selection enables you to select specific entities such as that you are willing to select in graphics window . It is available on right end of message bar .

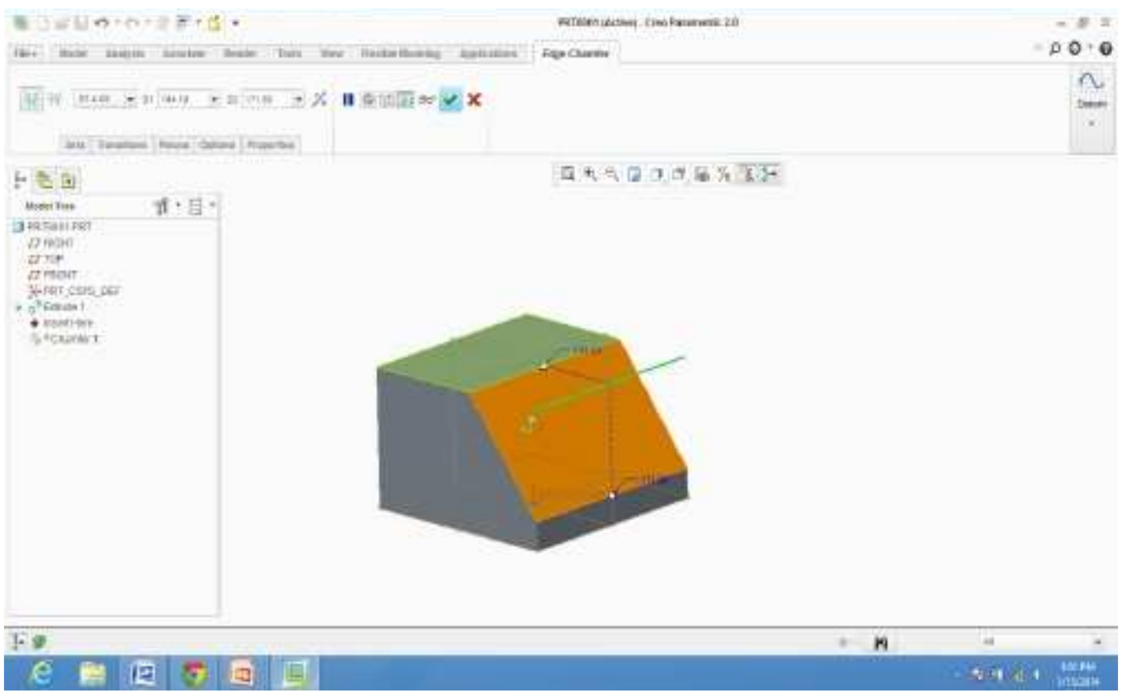
CHAMFERS

Chamfers add or remove material by creating a beveled surface between adjacent surfaces and edges .

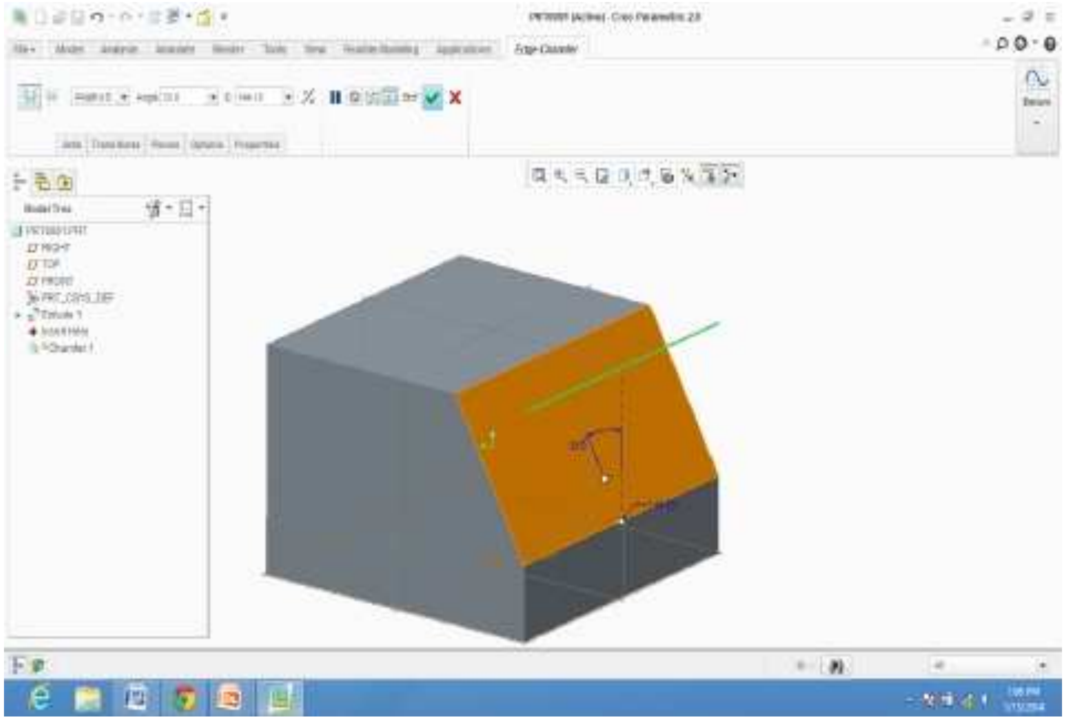
- Common distance(DxD): Size of chamfer is defined by one dimension



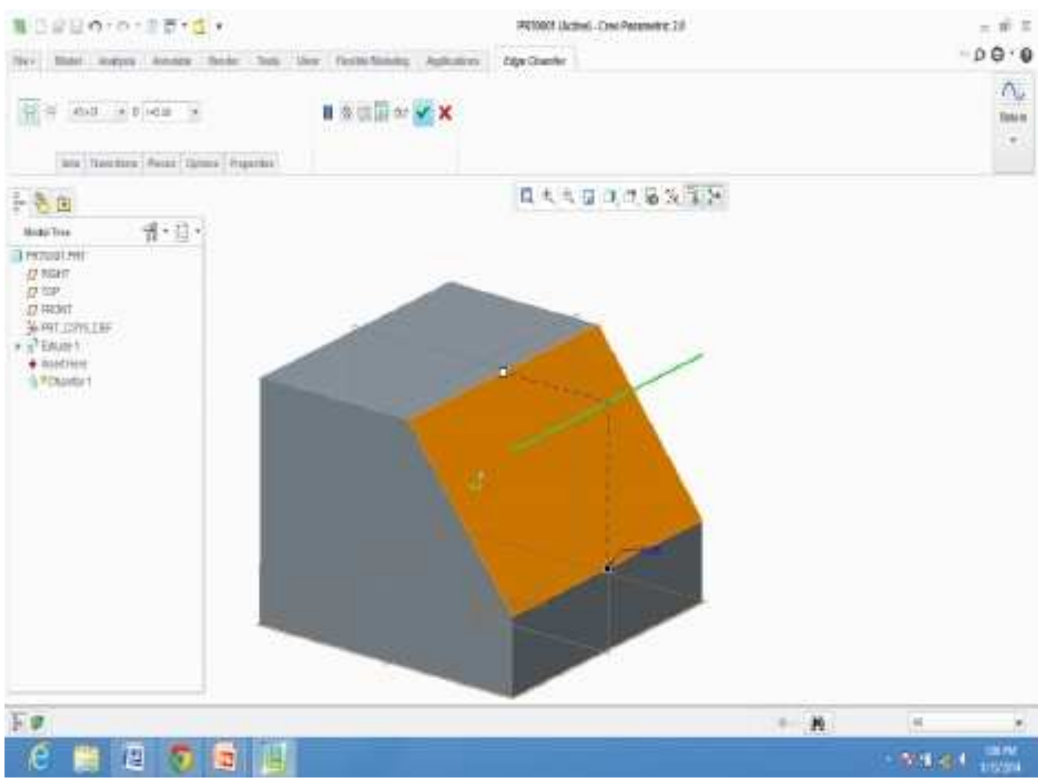
- Variable distance(D1xD2): Size of chamfer is defined by two dimensions



- Angle X distance(AxD): Size of chamfer is defined by linear and angular dimensions



- 45 X distance(45xD):Size of dimensions is defined by a linear dimension and an angle of 45 degree .



CHAPTER 5
CREATING REVOLVES AND ROUNDS

It is an Addition or Removal of Material around the axis

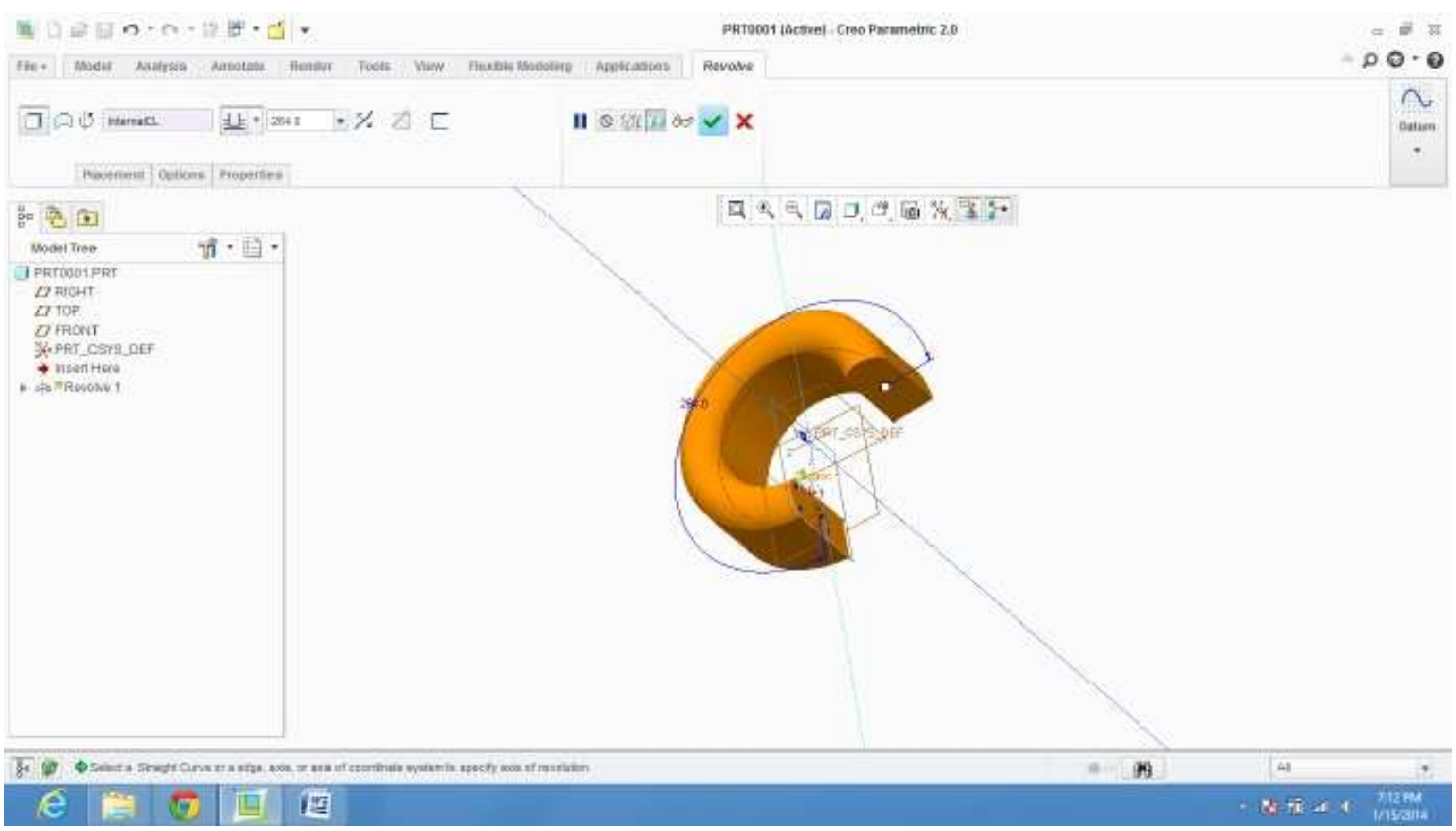
- ***Geometric Center line*** is compulsory for ***Revolution.***

Steps involved :

Step 1: Go to revolve>placement>define>select plane

Step 2: Define axis of revolution

Step 3: Create cross section



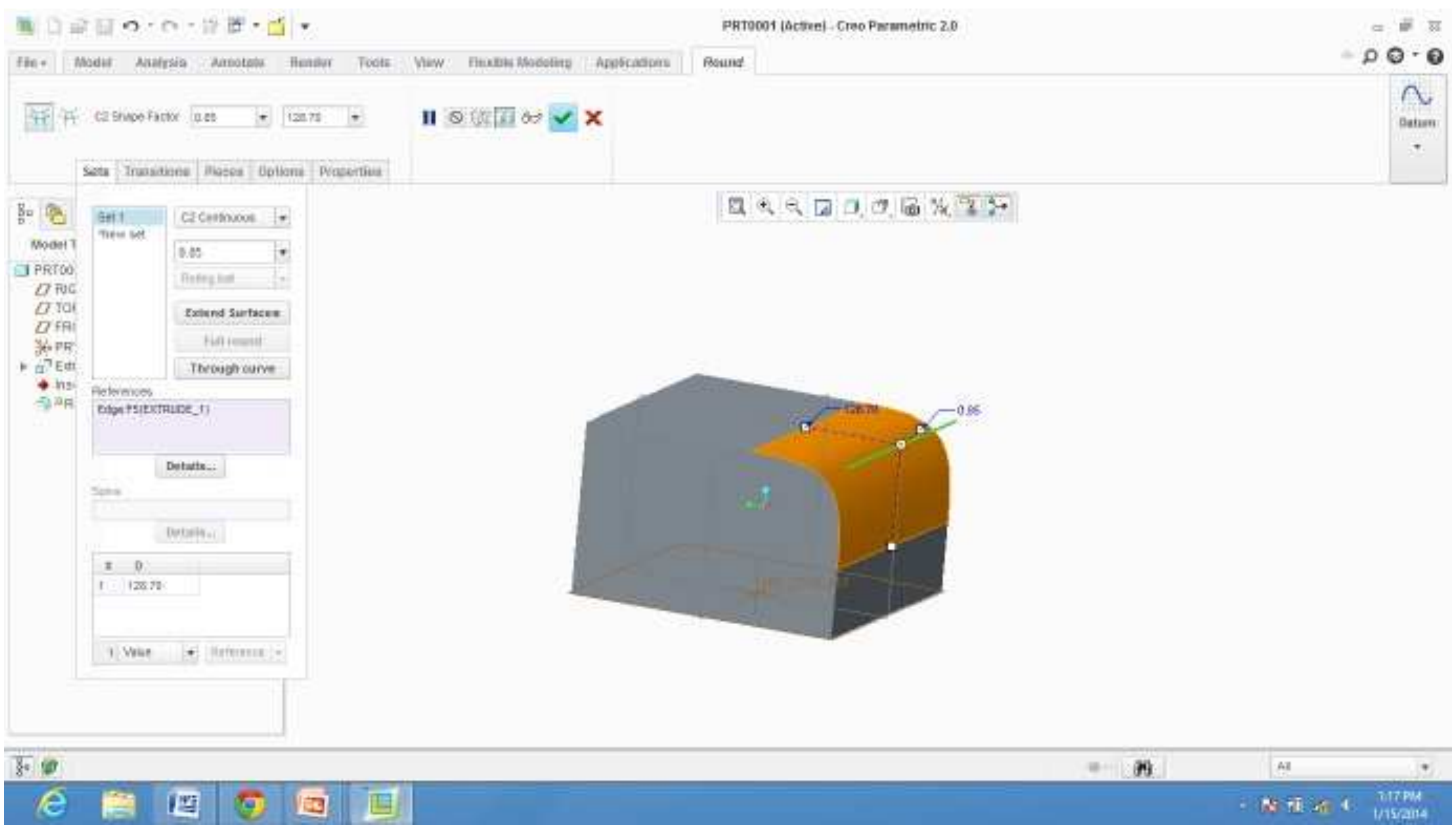
Note:

1> To revolve as a solid section must be closed
2> To revolve as a surface section must be open .

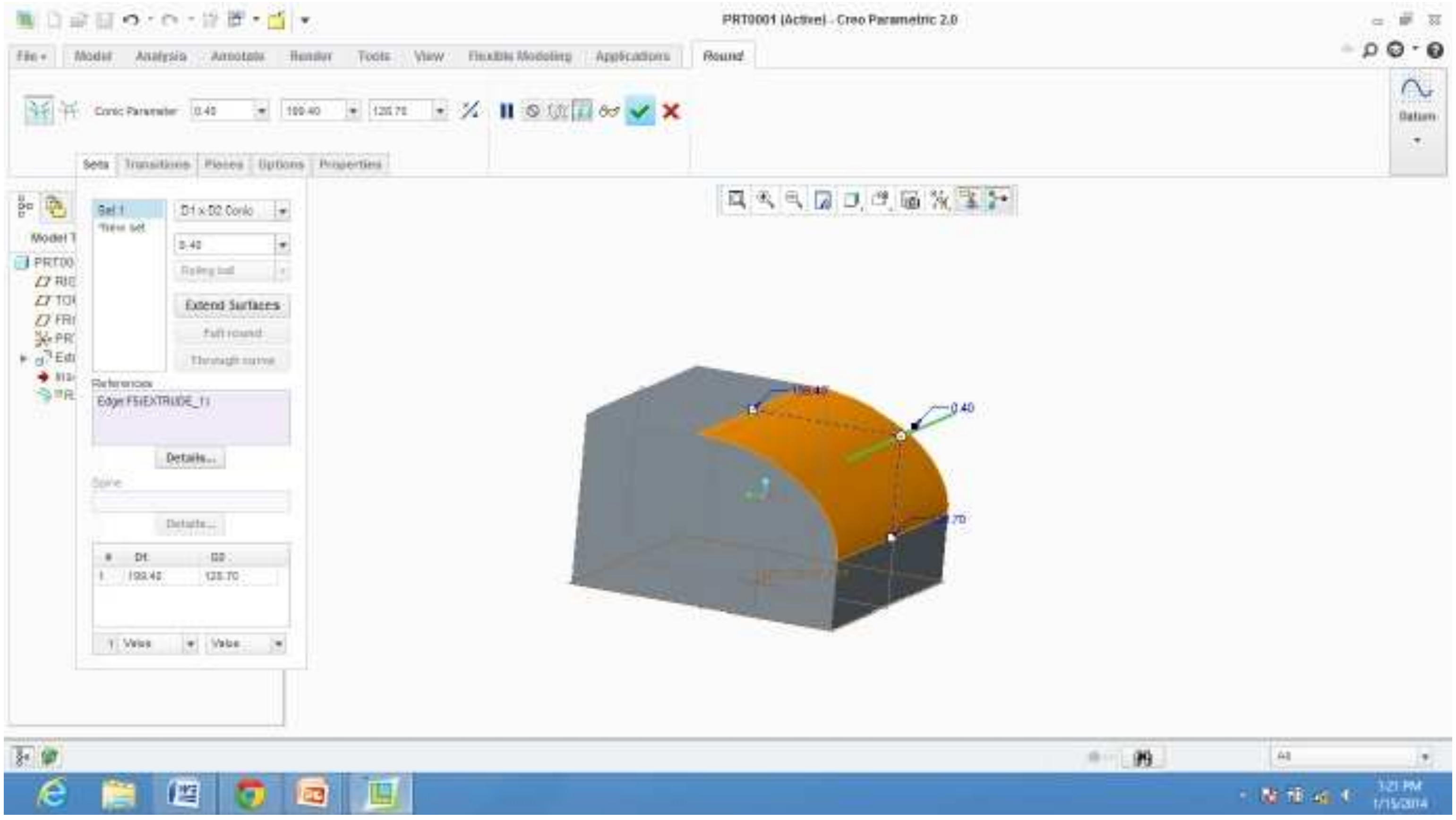
ROUNDS

Types of rounds are :

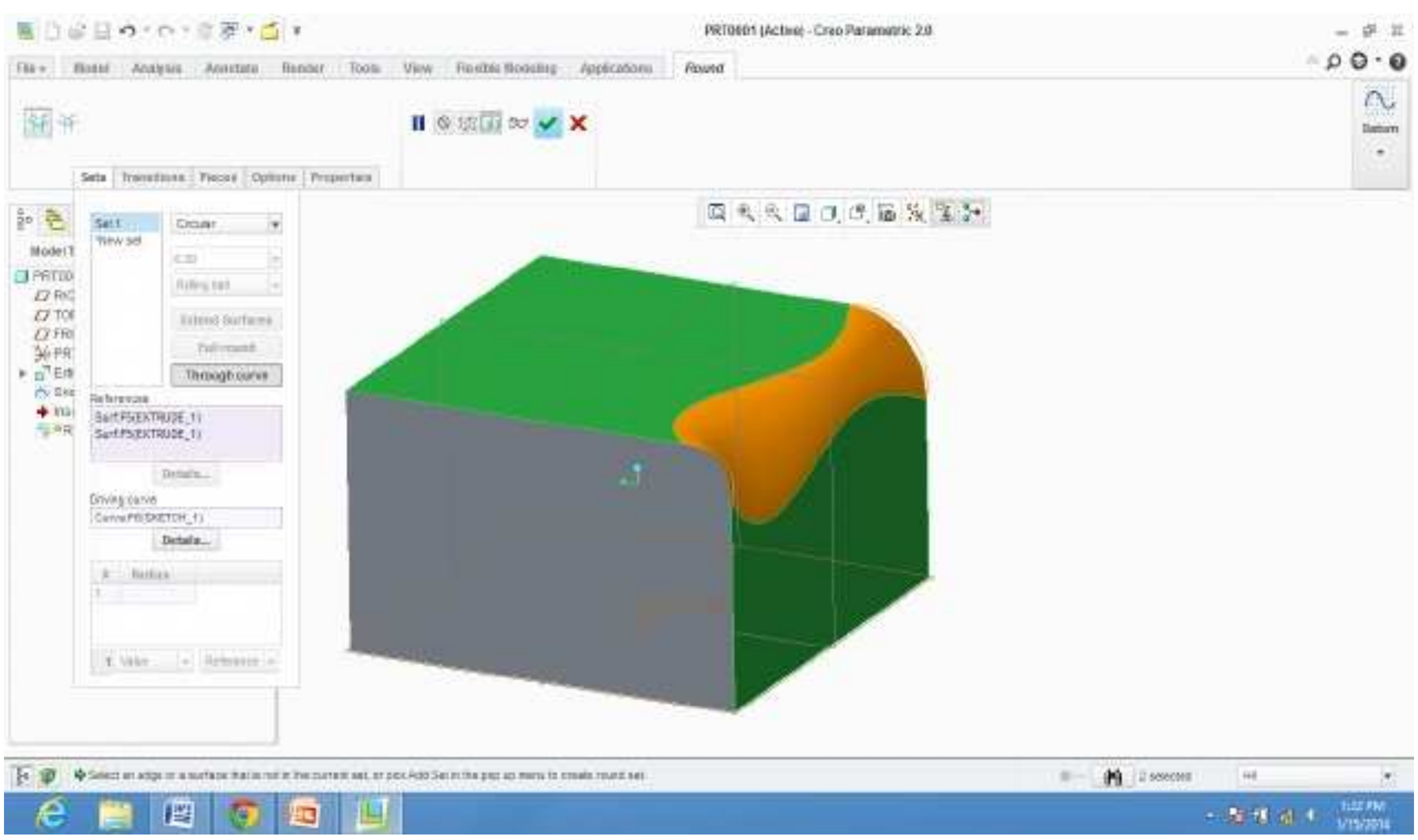
1> C2 continuous : Manage conic value and radius



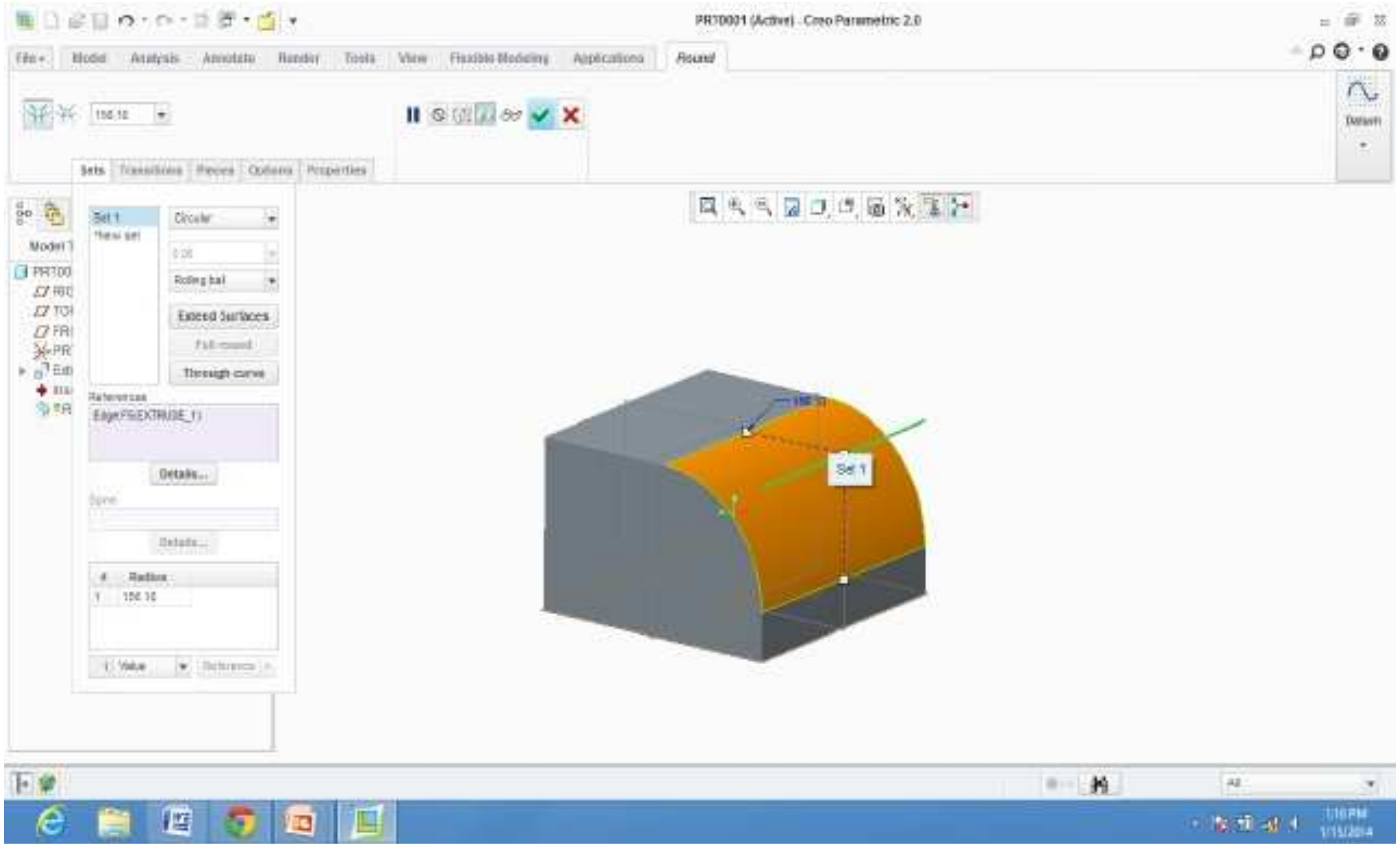
2> D1*D2 conic: Manage radius for both surfaces and conic value also .



3> Through curve : Select 2 surfaces and curve



4> Circular : Manage radius only for both surfaces .


PRT0001 (Active) - Creo Parametric 2.0
File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Round
Datum
Sets
Transitions
Pieces
Options
Properties
Set 1
*New set
Circular
Rolling ball
Extend Surfaces
Full round
Through curve
References
Details...
Radius
Value
Reference
Set 1

CHAPTER 6
UNDERSTANDING DATUMS

Datum Features are not having the mass properties but it is used to give the reference.

There are 5 types of datum features :

1> Datum planes

2> Datum points

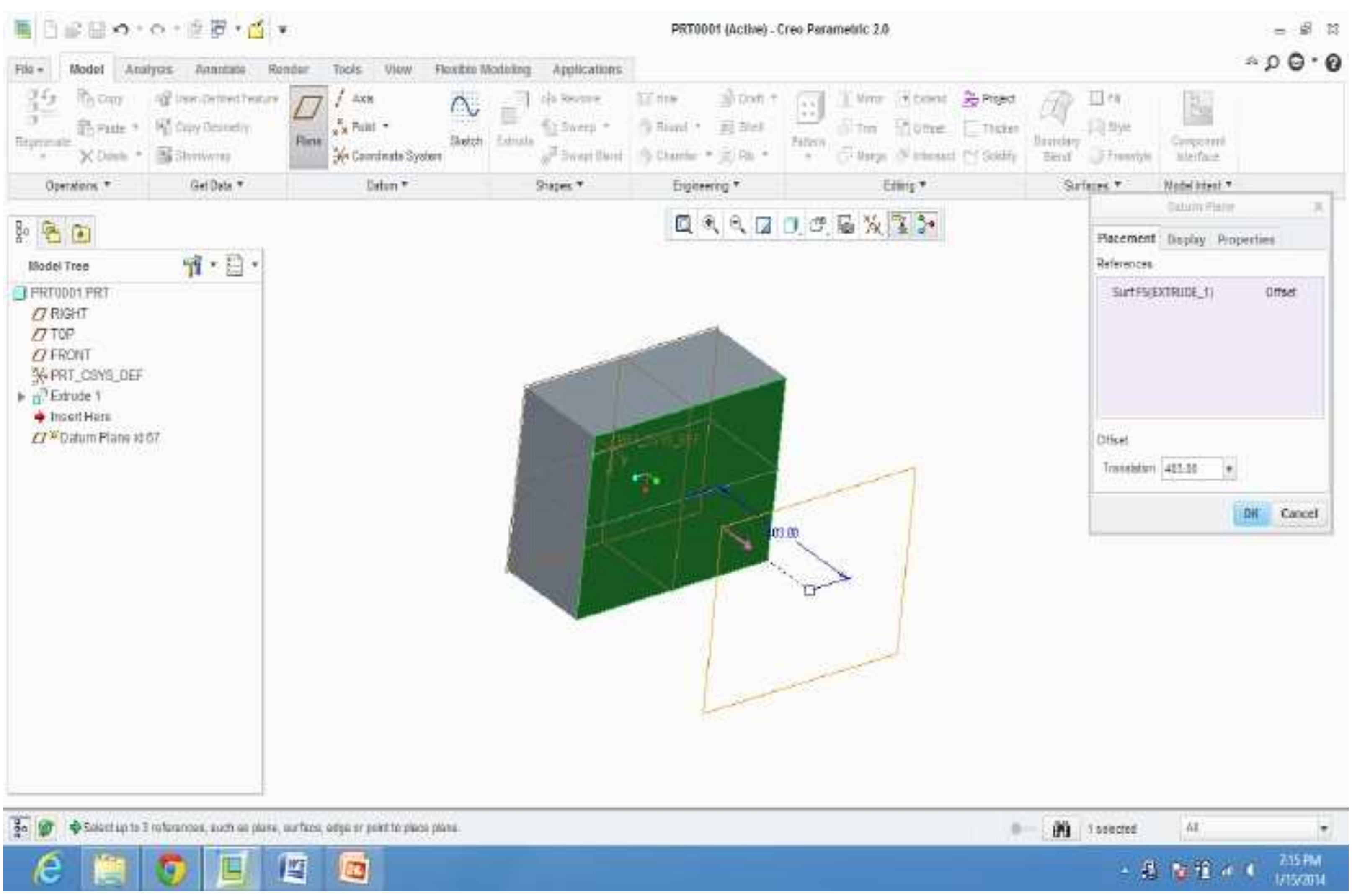
3> Datum axis

4> Datum coordinate systems

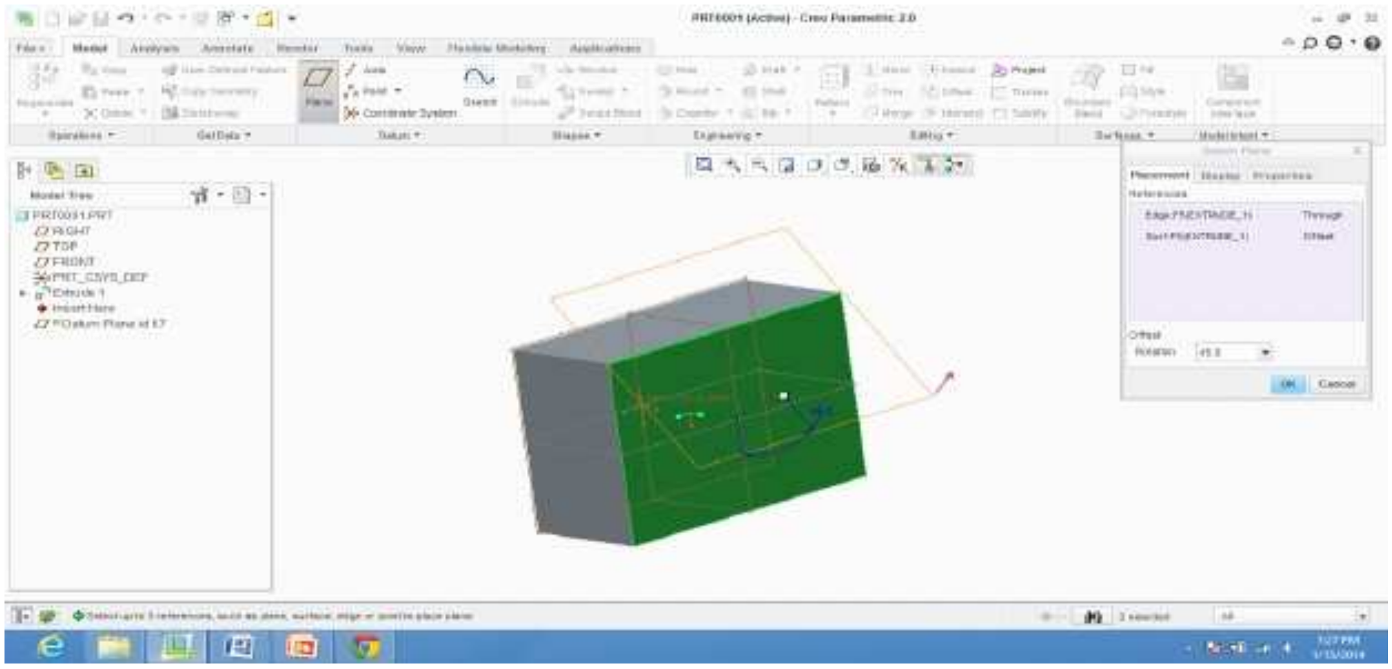
5> Datum curves

1> TYPES OF DATUM PLANES

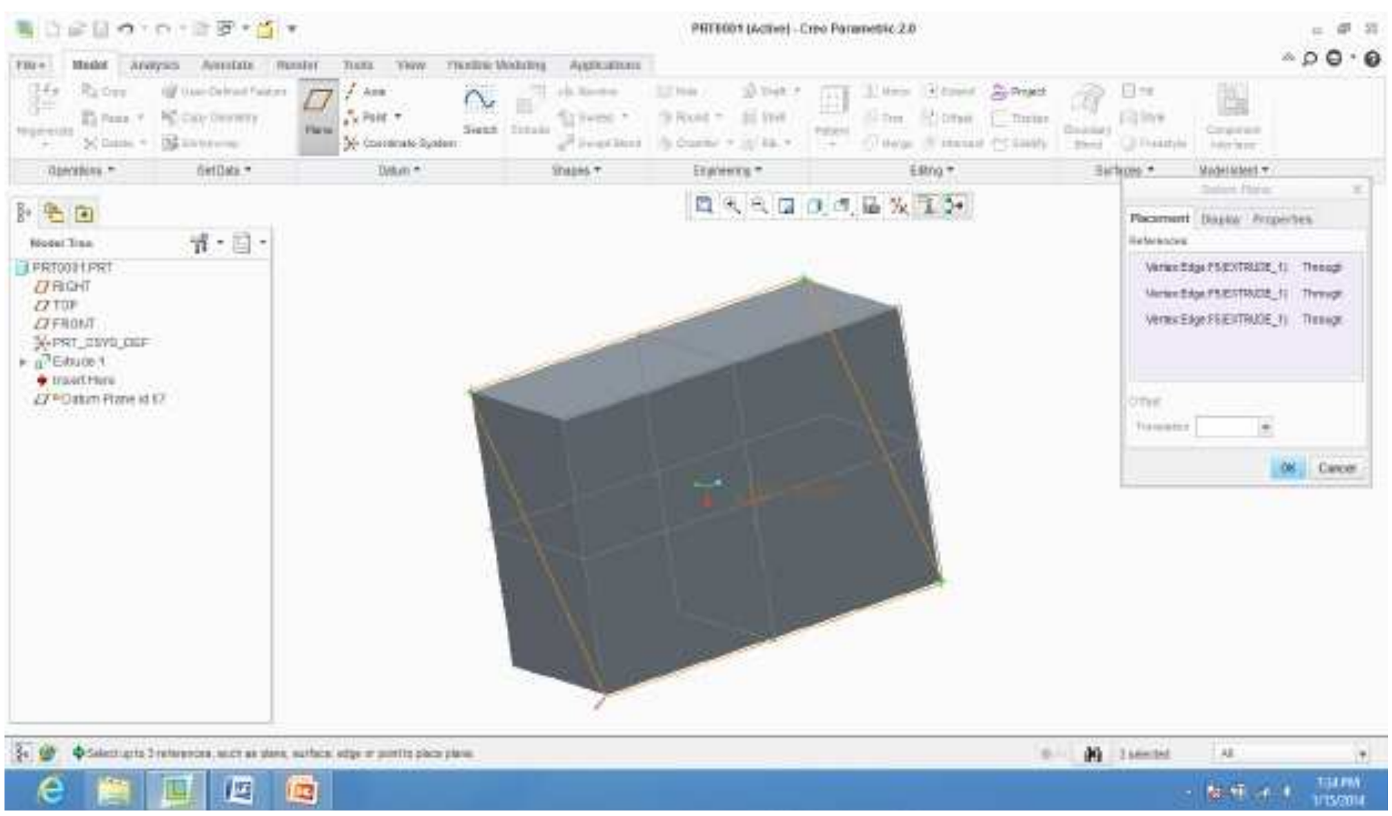
Case 1: PARALLEL OFFSET : Either surface or default planes can be selected .(Front , right , top) . Specify distance from plane or surface .



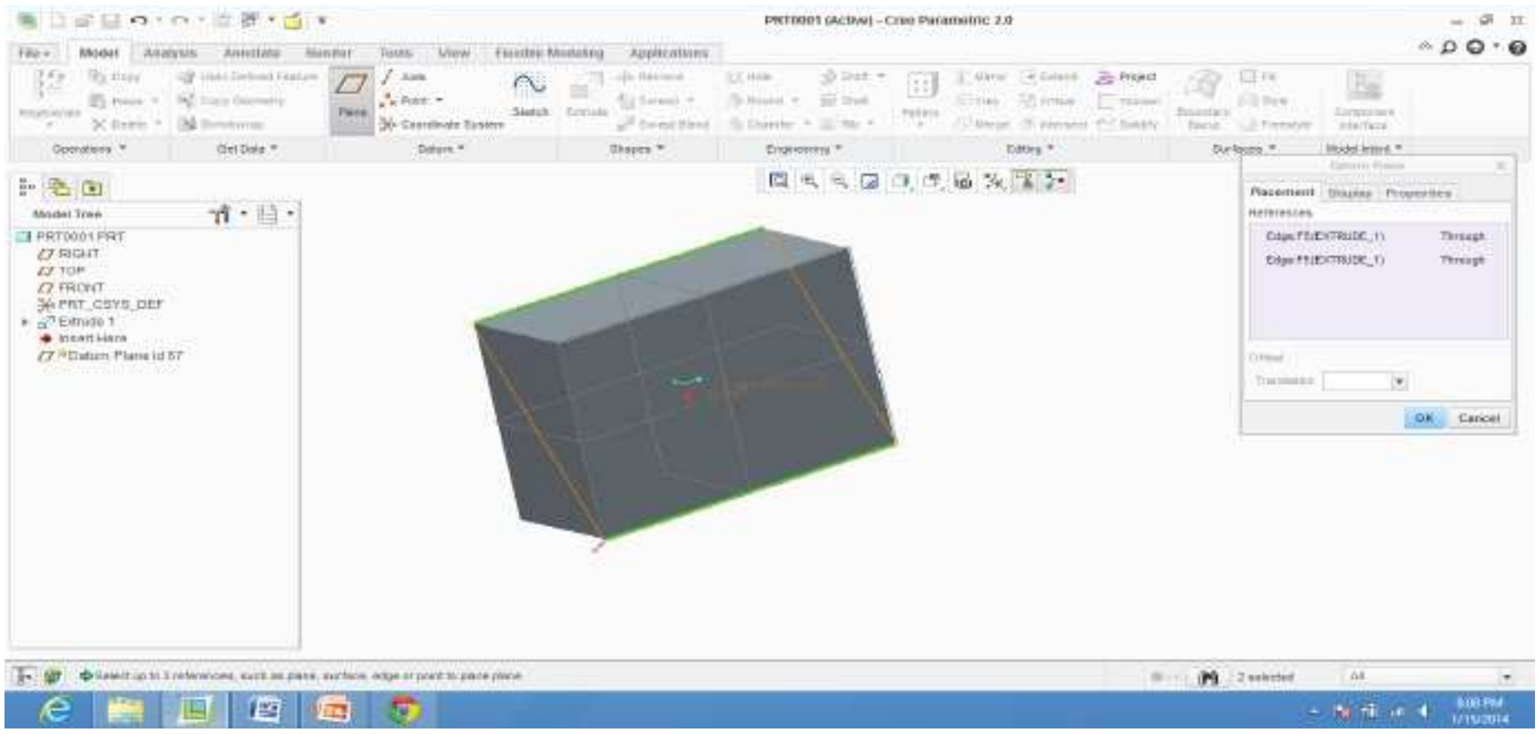
Case 2: Plane passing through an edge/curve and angle from surface/ plane . Select edge / curve and ctrl+surface from which we need to specify angle .



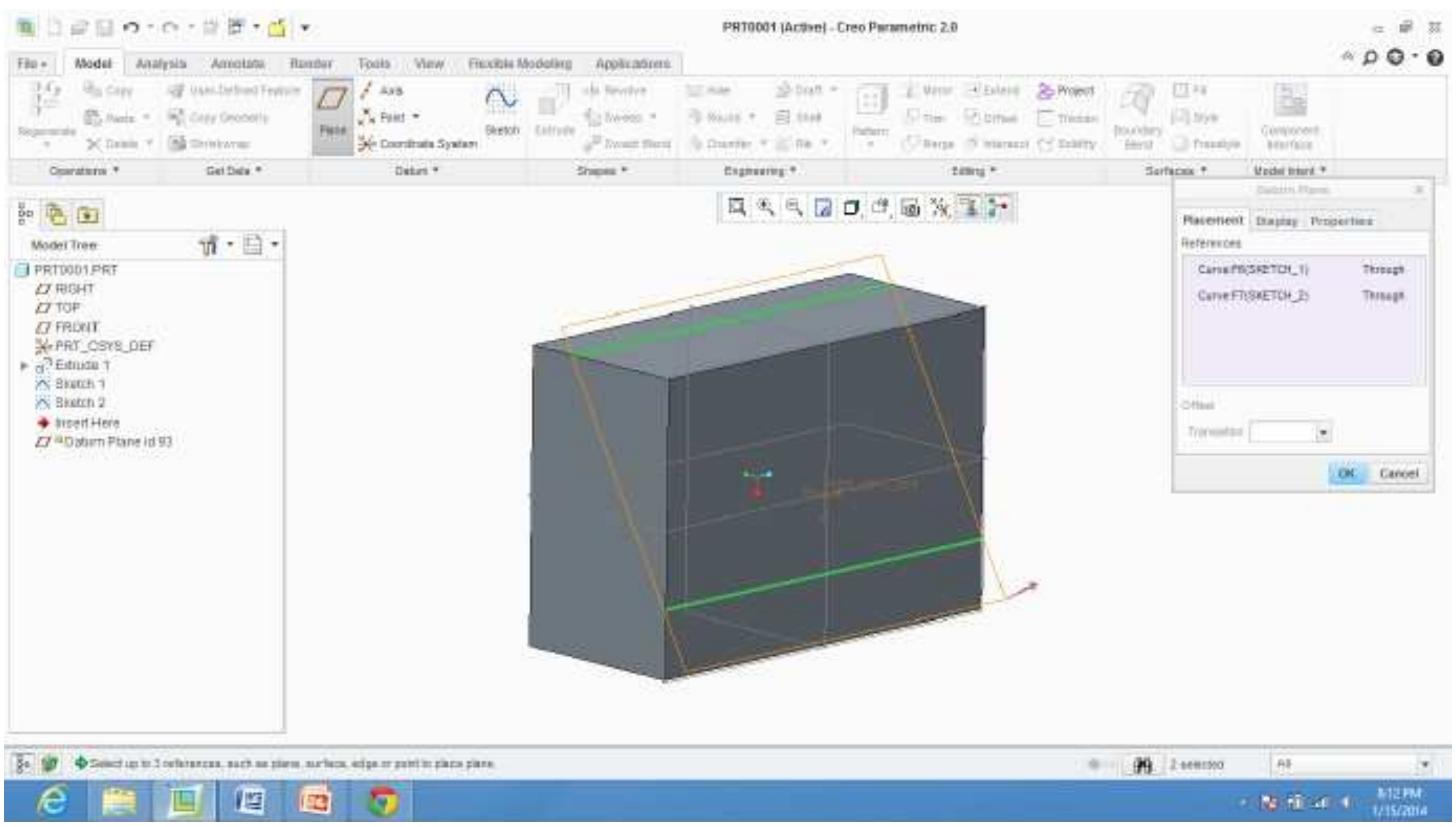
Case 3: Plane passing through 3 points .



Case 4: Plane passing through 2 edges, 2 curves



Plane passing through 2 curves .



Uses of datum planes :-

1> Sketching plane and reference plane for sketching

2> Dimensioning and alignment references in the sketch

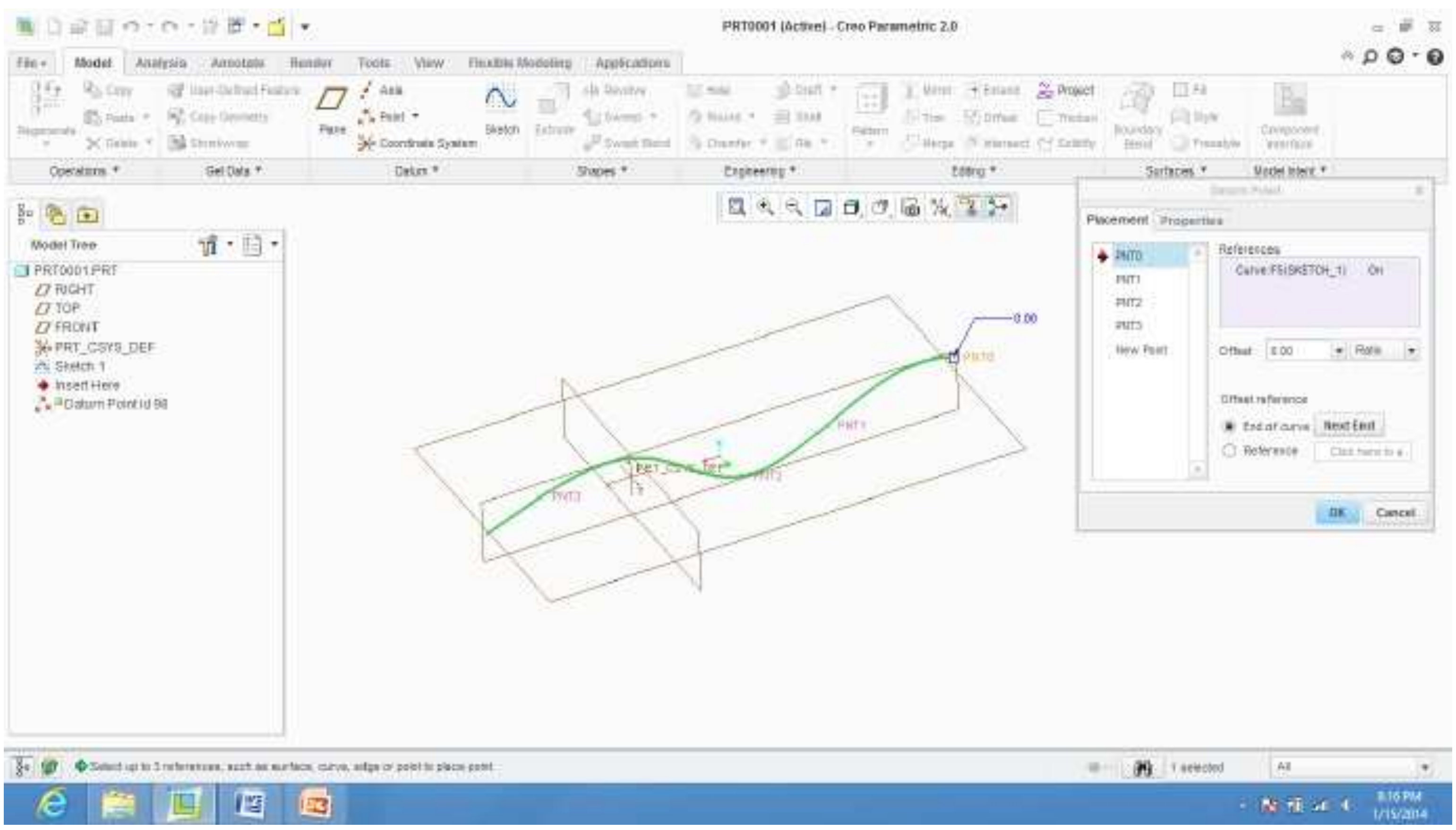
3> Feature depth references (to selected)

4> Creating cross sections

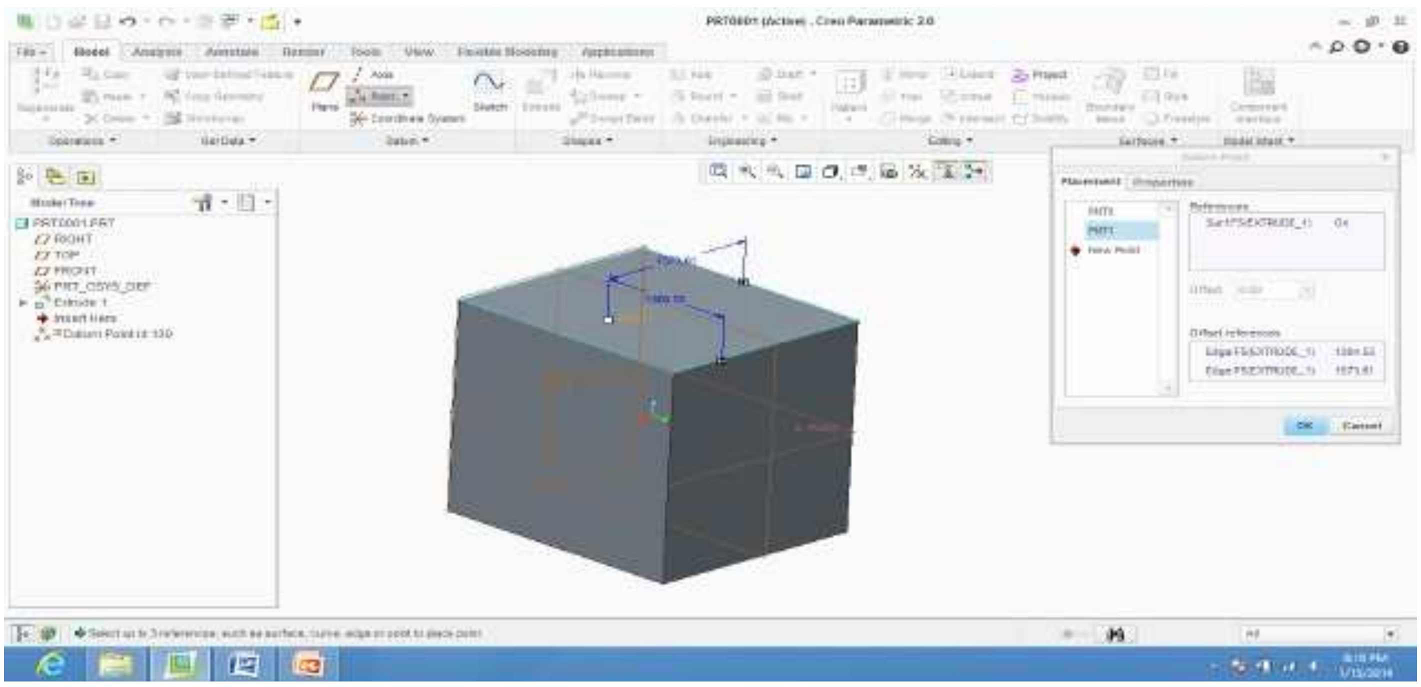
5> Reference plane for mirror tool

DATUM POINTS

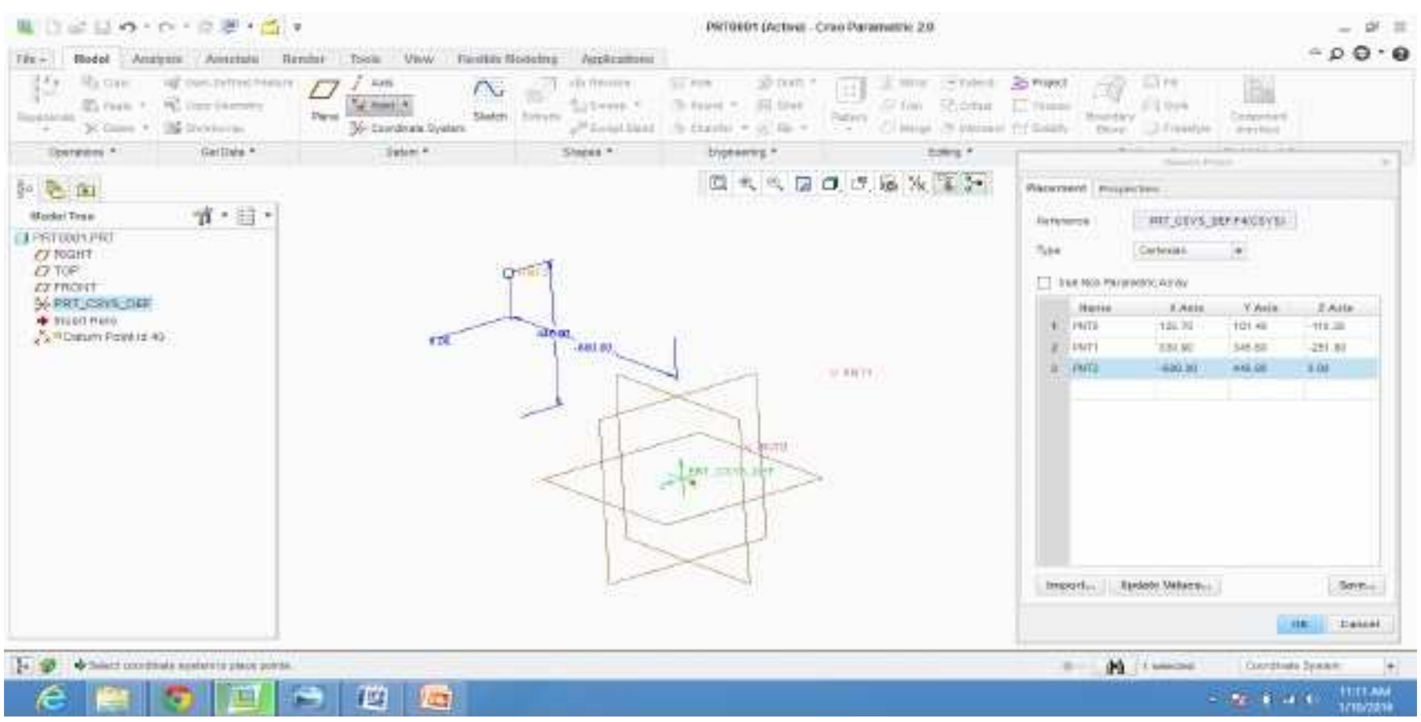
1>On curve : click on point > click on curve > manage location of point by ratio or by real (distance from start point)



2> On surface : Goto point > click on surface > by drag handle manage distance from orthogonal edges/surfaces.



3> Offset coordinate system : Go to point > offset coordinate system> click on part coordinate system > click on row> assign values of x,y,z > click on next row > again assign values for x,y,z coordinates



USES OF DATUM POINTS

1. To create datum planes and axes.

2. To associate note in the drawings and attach datum targets.

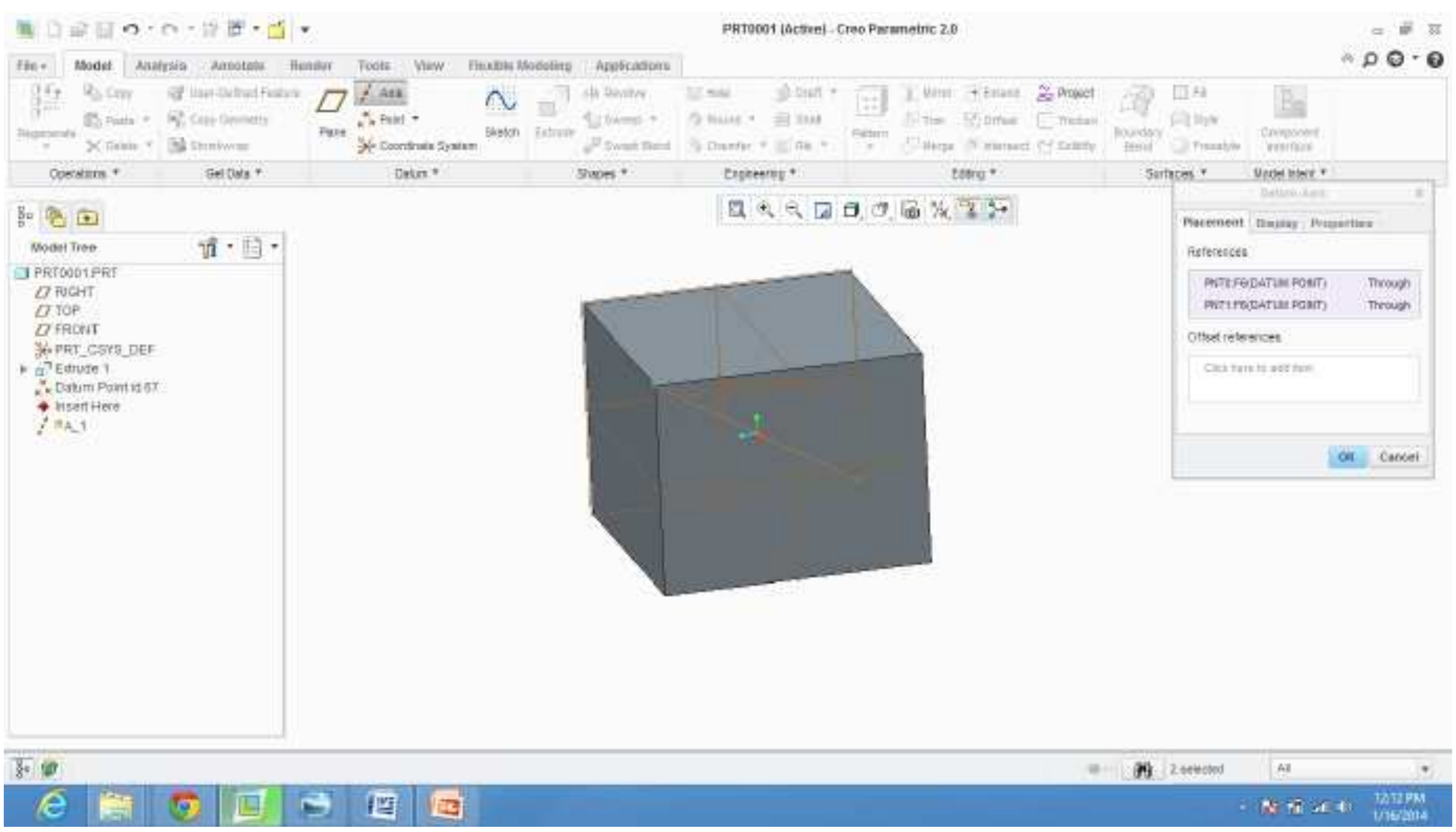
3. To create coordinate system.

4. To specify point loads for mesh generation.

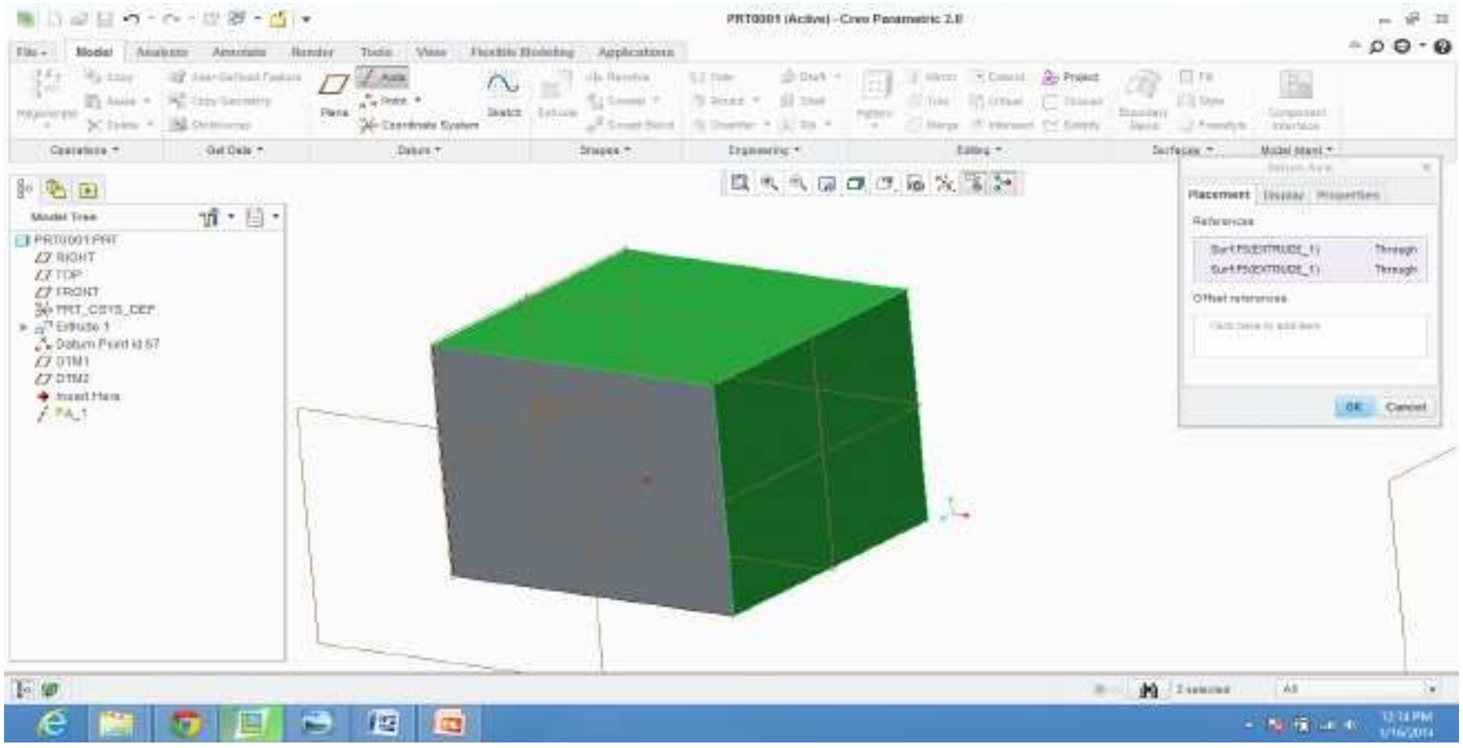
5. To create pipe features.

DATUM AXIS

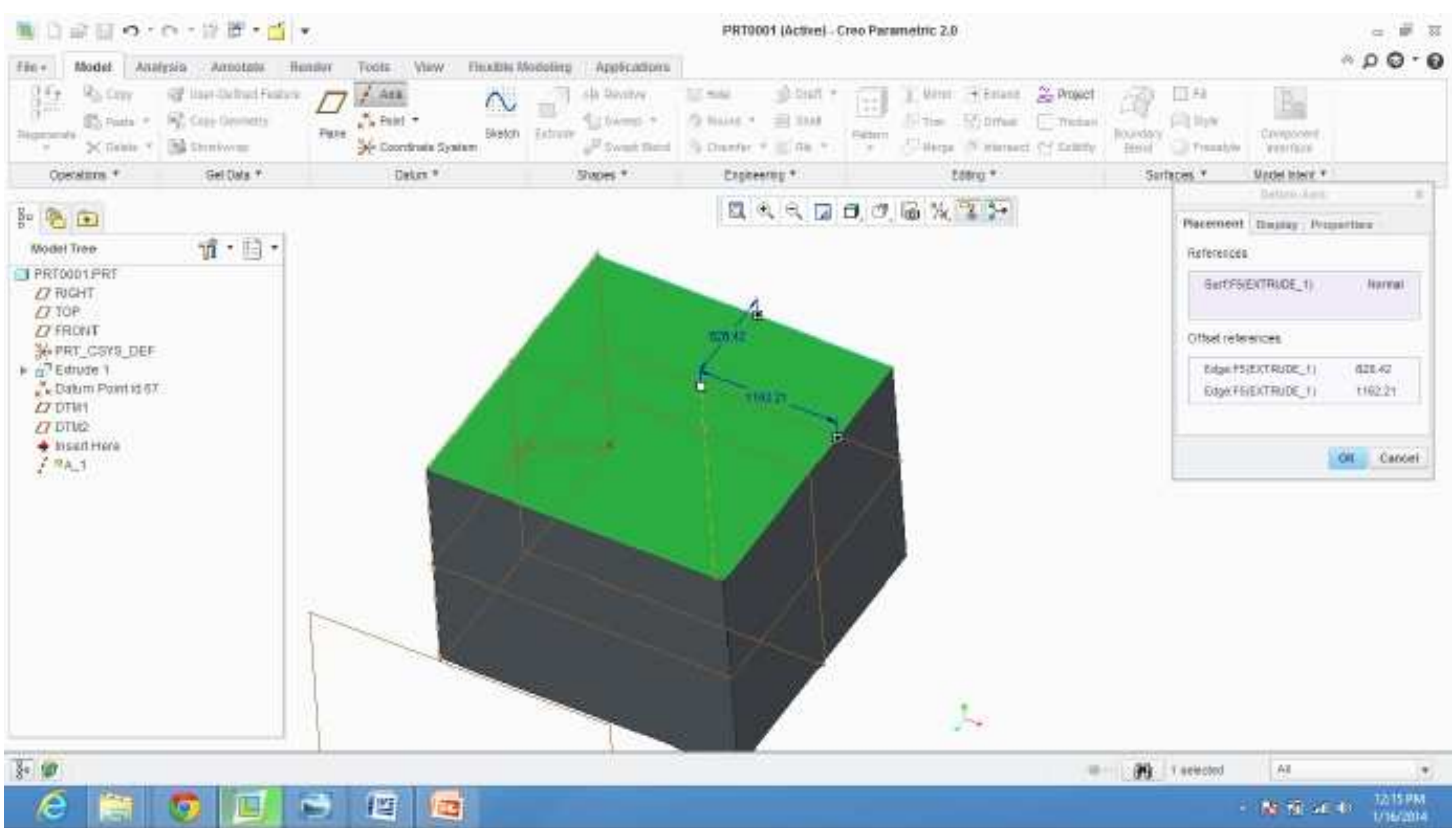
1> With 2 points



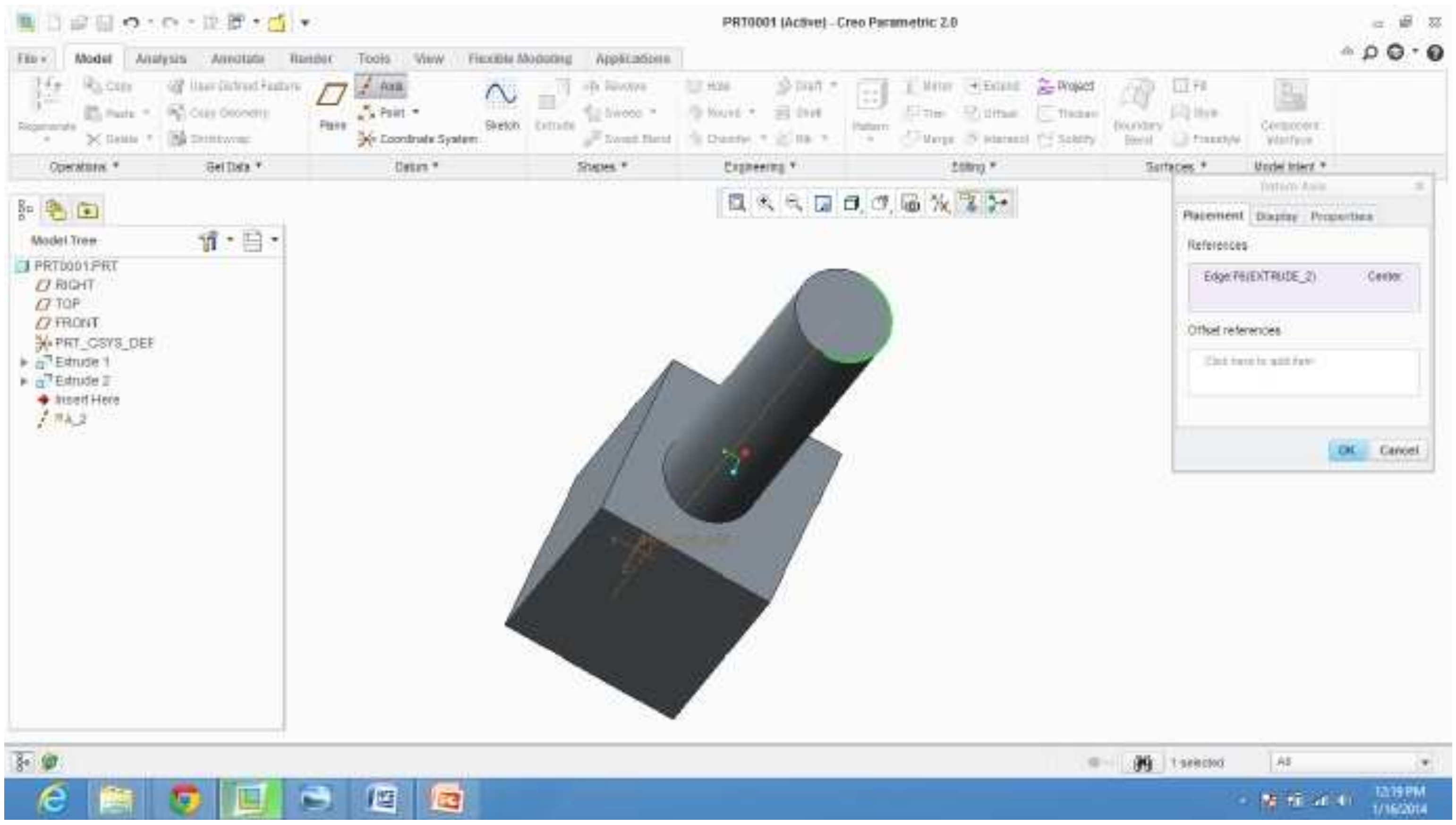
2> With 2 planes/surfaces : Select 2 adjacent surfaces . Axis will be created at intersection



3> On surfaces : Click on surface and manage distance from 2 orthogonal edges/surfaces by using drag handles .



4> Center of curved surface: Click on curved edge or surface to create a datum axis passing through center of cylindrical feature .



Uses of datum axis:

1> Creating coaxial holes.

2>Centerlines on drawings.

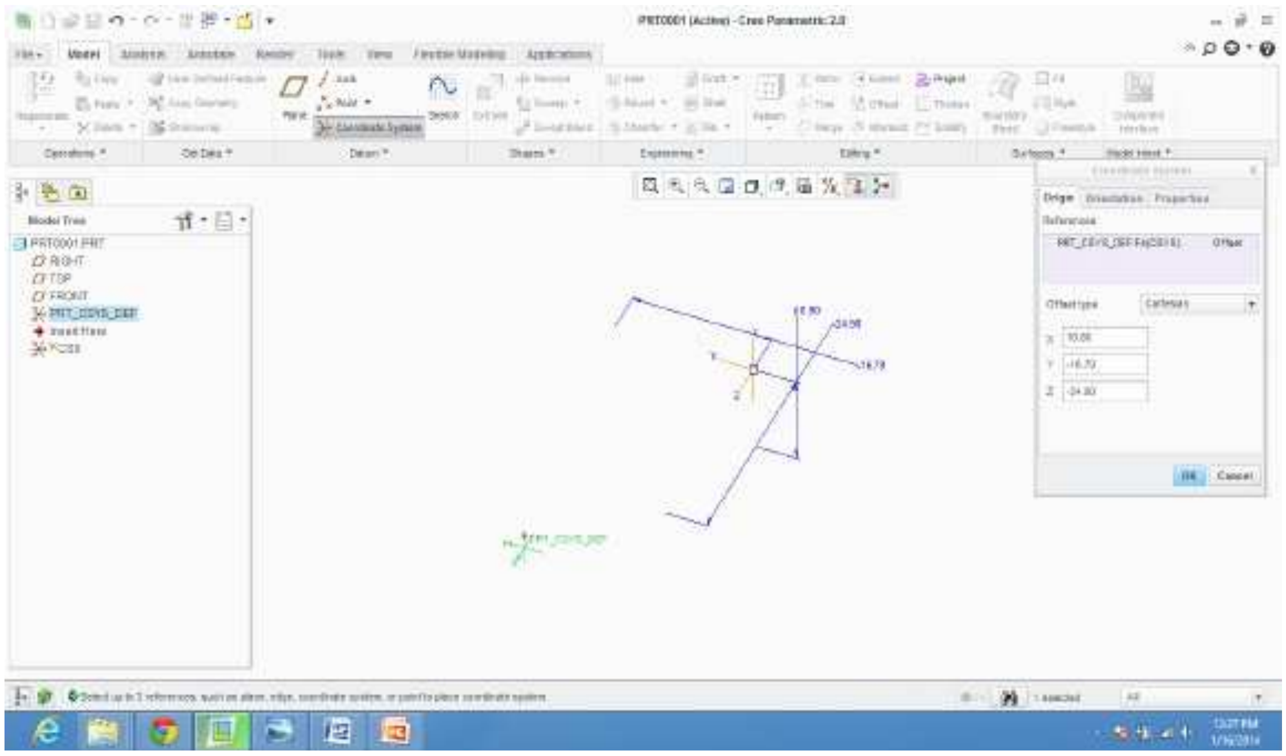
3>To indicate symmetry on drawings.

4>Geometric tolerances.

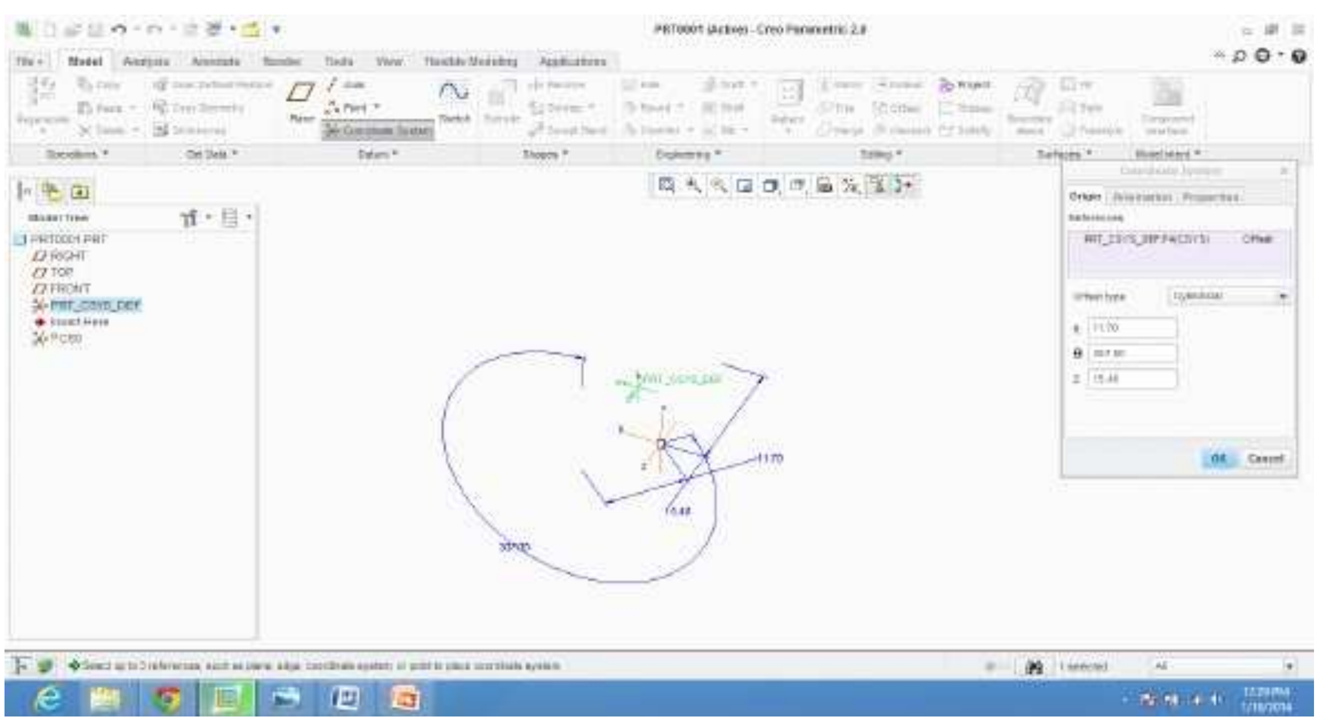
5>Assembly placement constraints.

DATUM COORDINATE SYSTEM

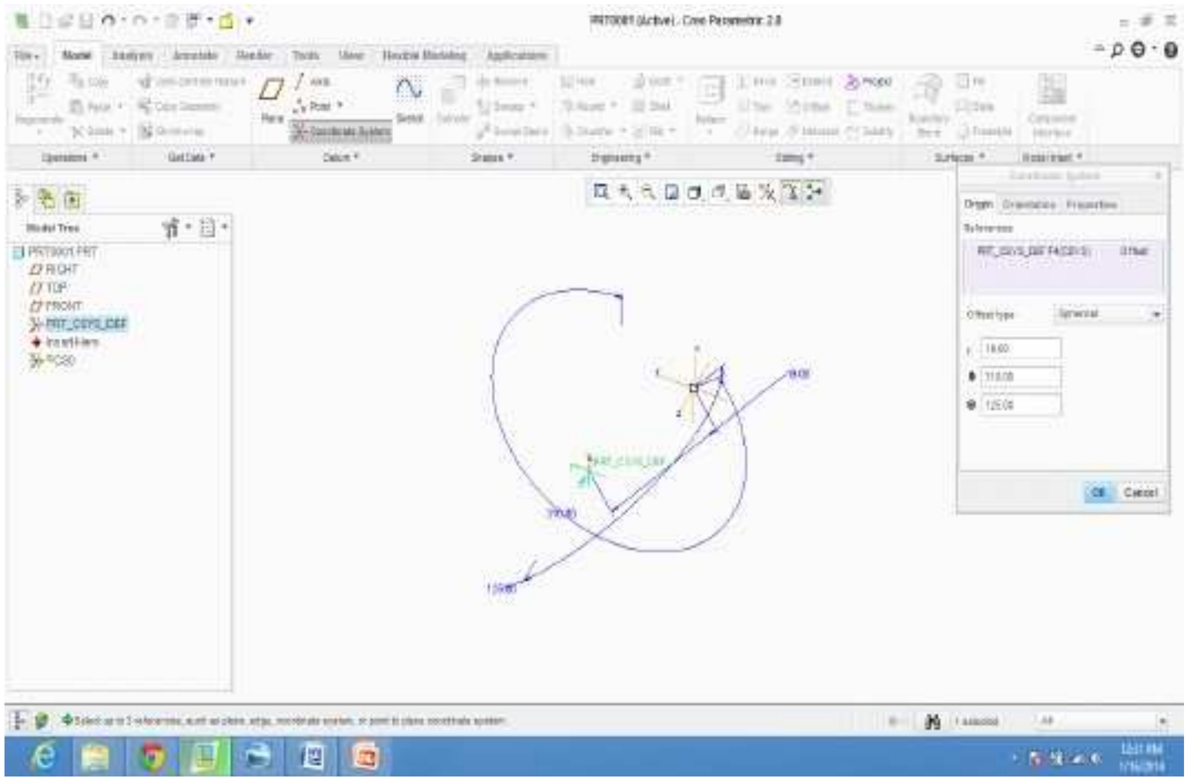
1>Cartisian-Created by defining x,y,z parameters . Click on datum coordinate system >click on part coordinate system > manage x,y,z values .



2> Cylindrical-Created by defining R, theta , Z parameters



3>Spherical-Created by defining r , theta , and phi parameters.



USE OF DATUM COORDINATE SYSTEM

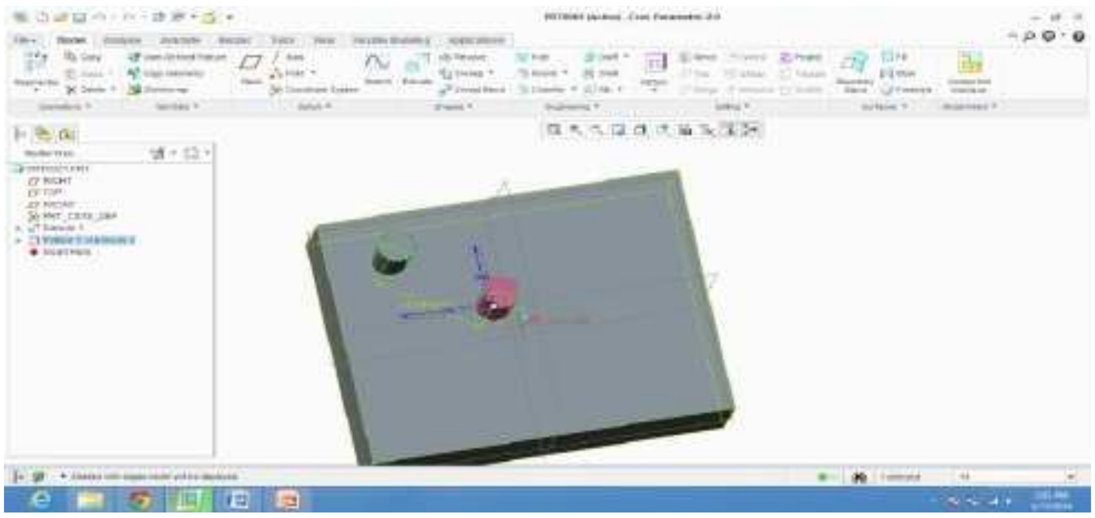
Datum coordinate system can be used as a modeling or assembly reference , as a basis for calculations , and for assembling components

CHAPTER 7
PATTERNS

There are 8 types of patterns in creo . They are dimension,direction,axis,fill,table , reference, curve, point.

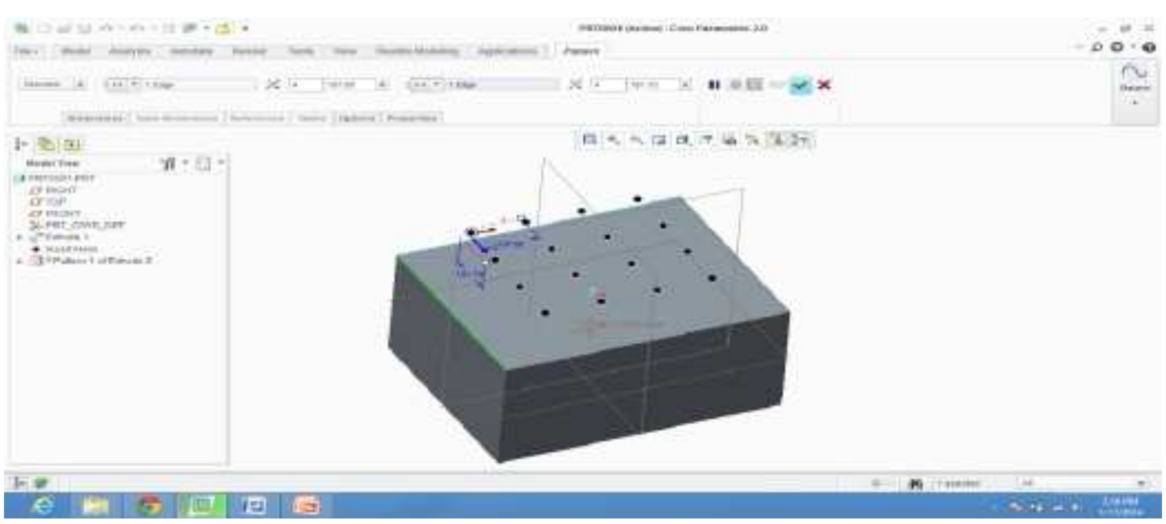
1> DIMENSION PATTERN

Select feature> go to pattern> select dimension pattern > go to dimensions tab > select 2 dimensions using ctrl from feature to vary > enter incremental value for dimensions> click done



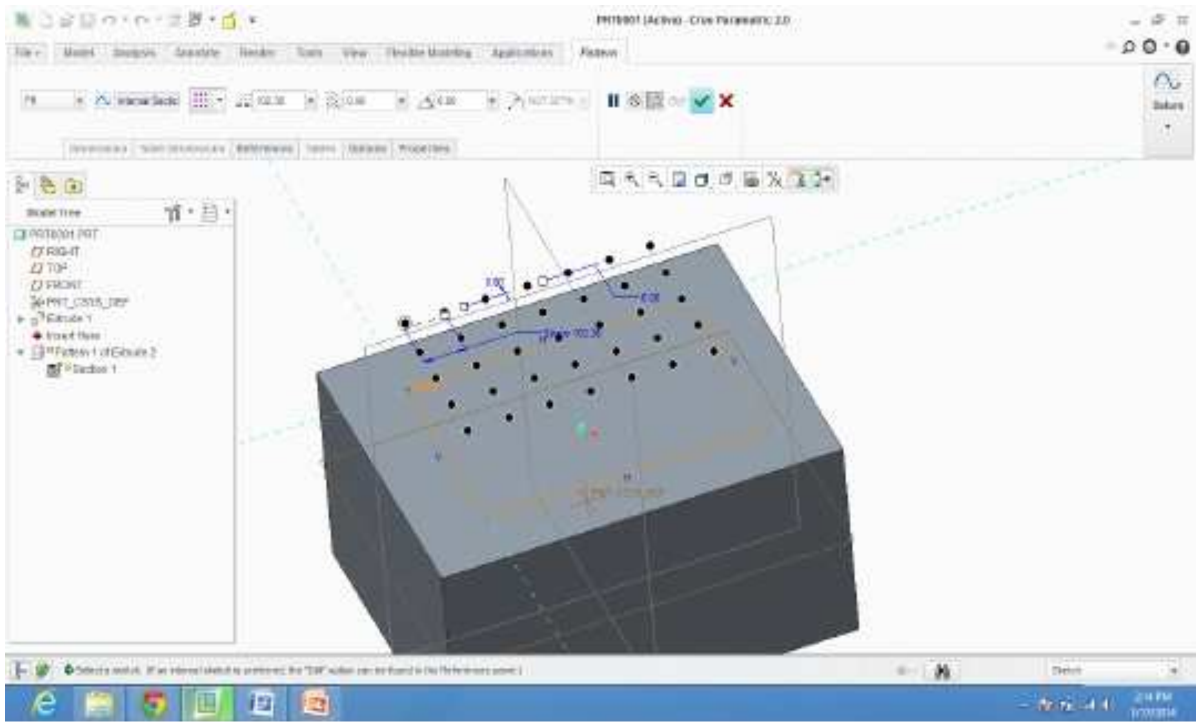
2> DIRECTION PATTERN

Select feature > go to pattern> select direction pattern > specify references(edges,planes,surfaces) in that direction
Manage spacing between copies and number of copies



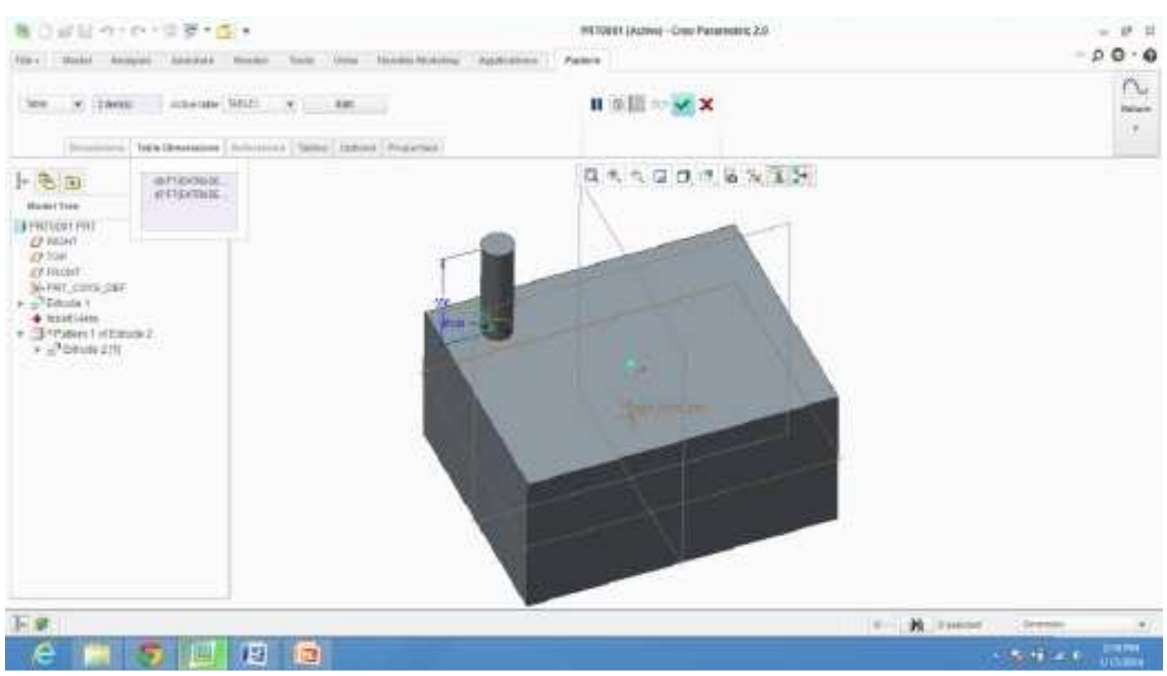
3> FILL PATTERN

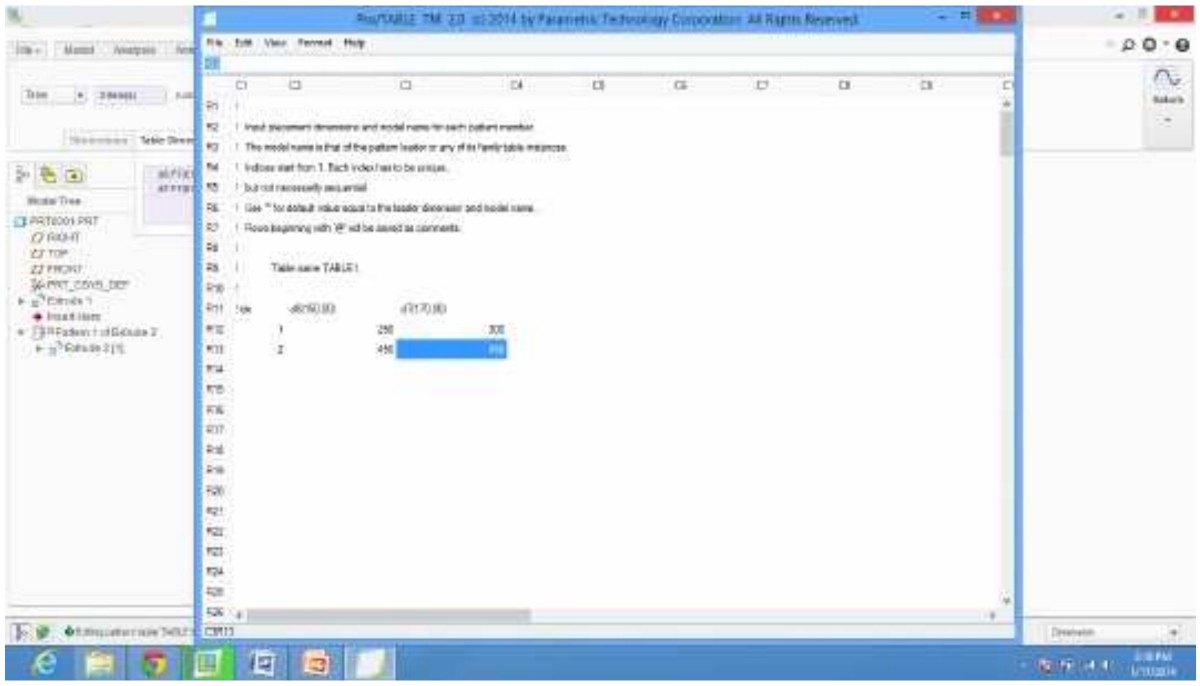
Select feature >go to pattern> select fill pattern> references > define>create sketch > manage spacing , angle and distance

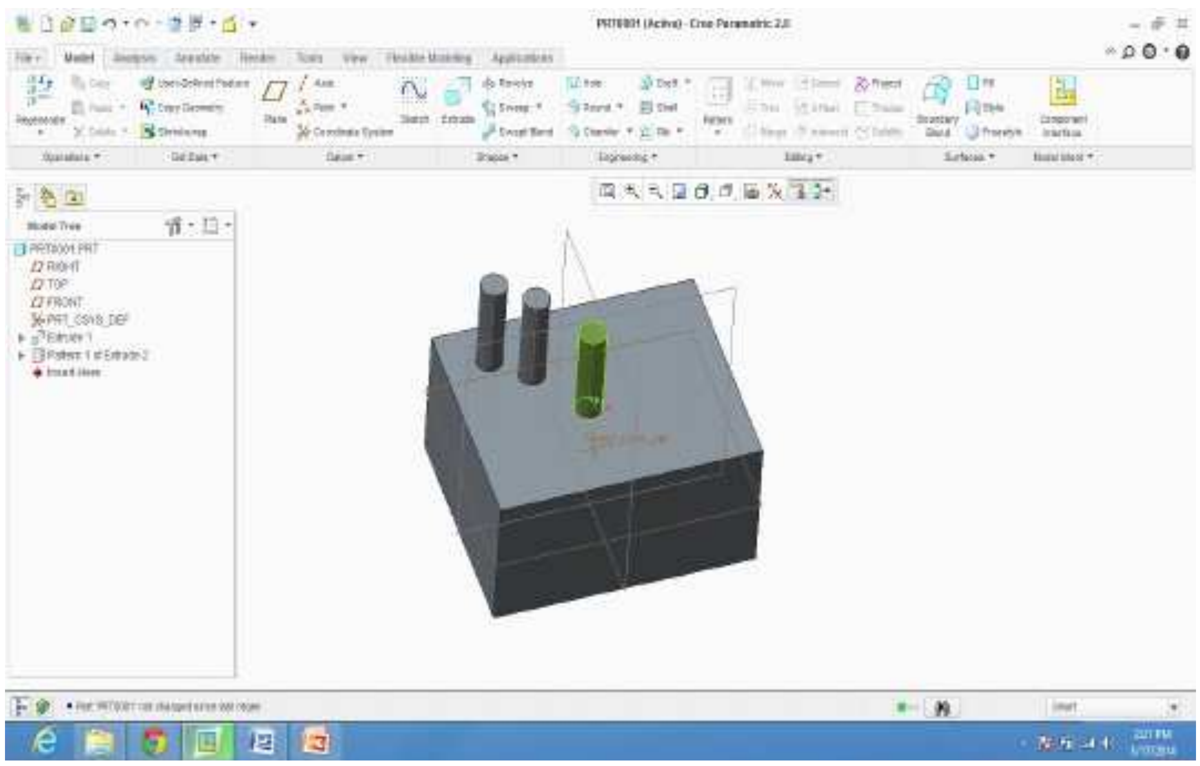


4> TABLE PATTERN

Select feature>pattern>table pattern>table dimensions > select multiple dimensions to vary using ctrl > click on edit > vary selected dimensions in table

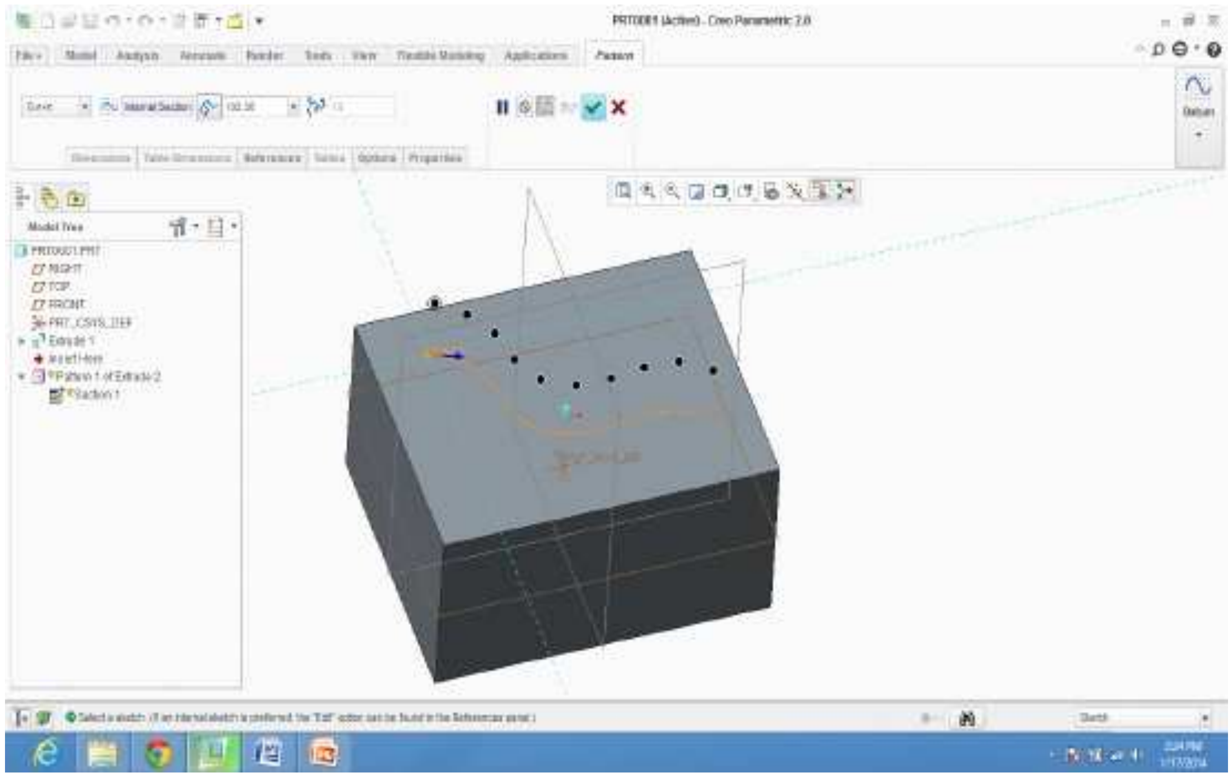


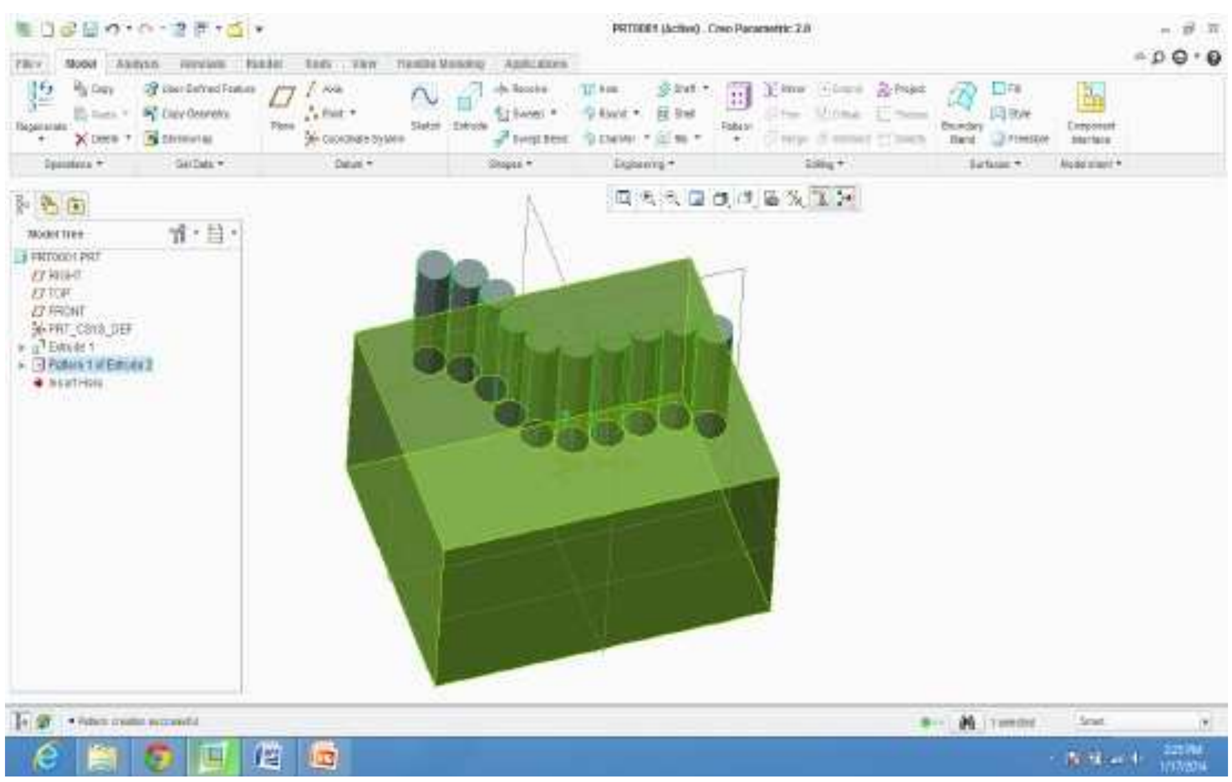




5> CURVE PATTERN

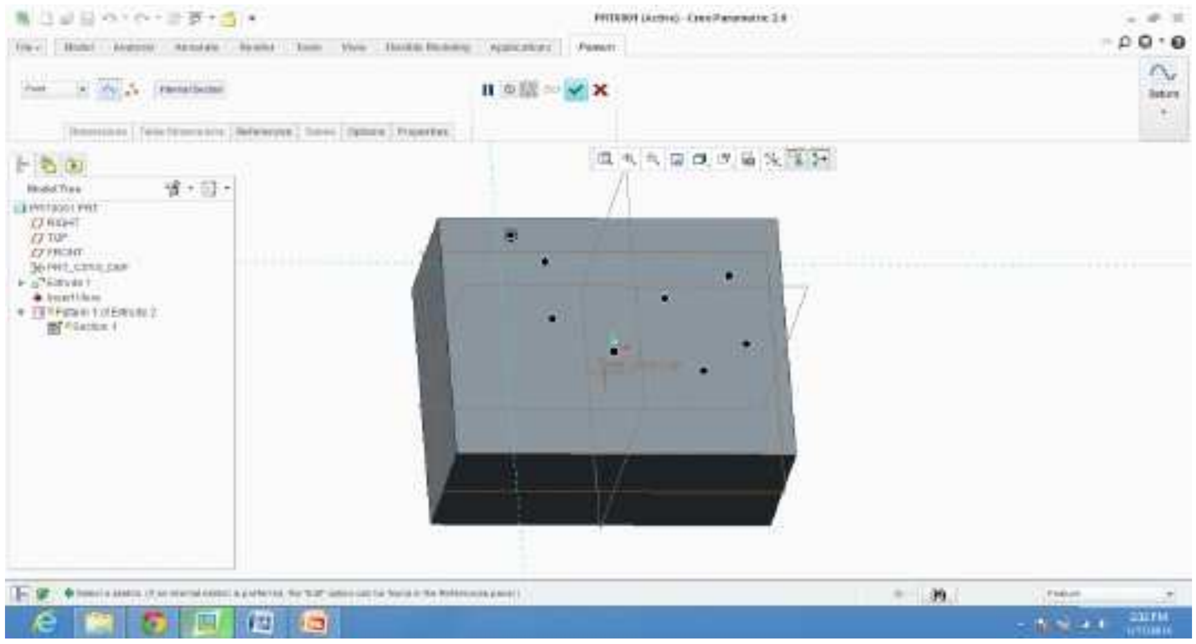
Select feature > goto pattern> select curve pattern> select curve > manage spacing between copies or total number of copies





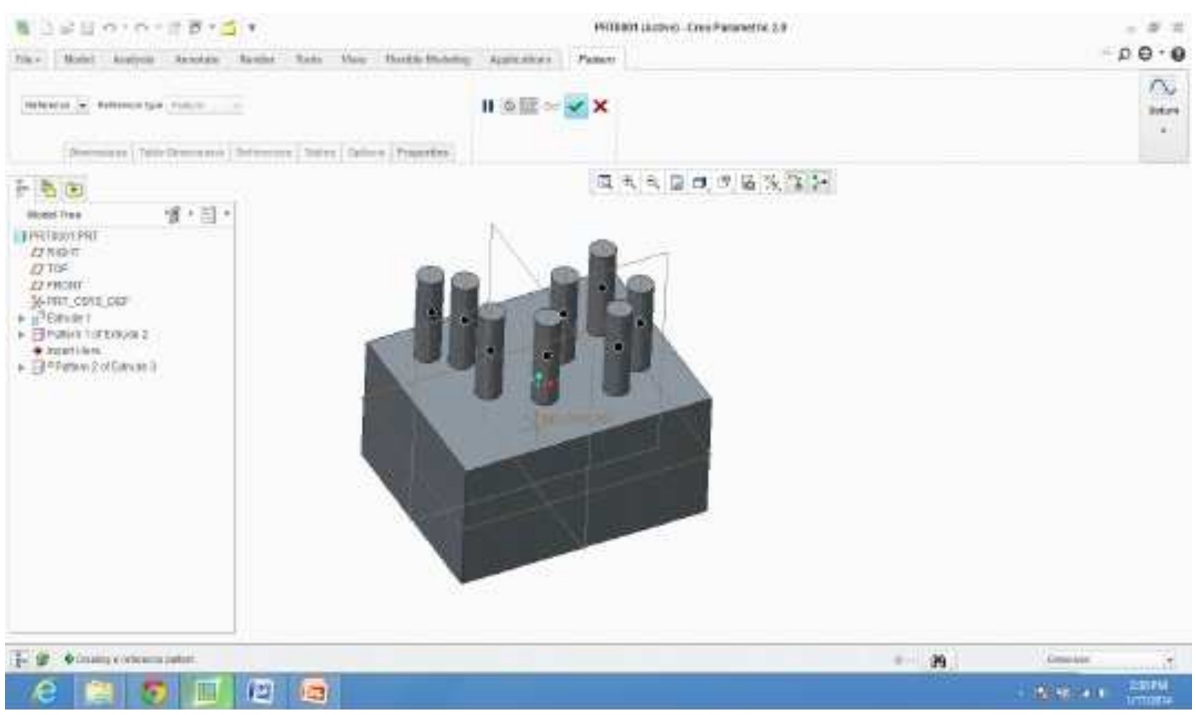
6> POINT PATTERN

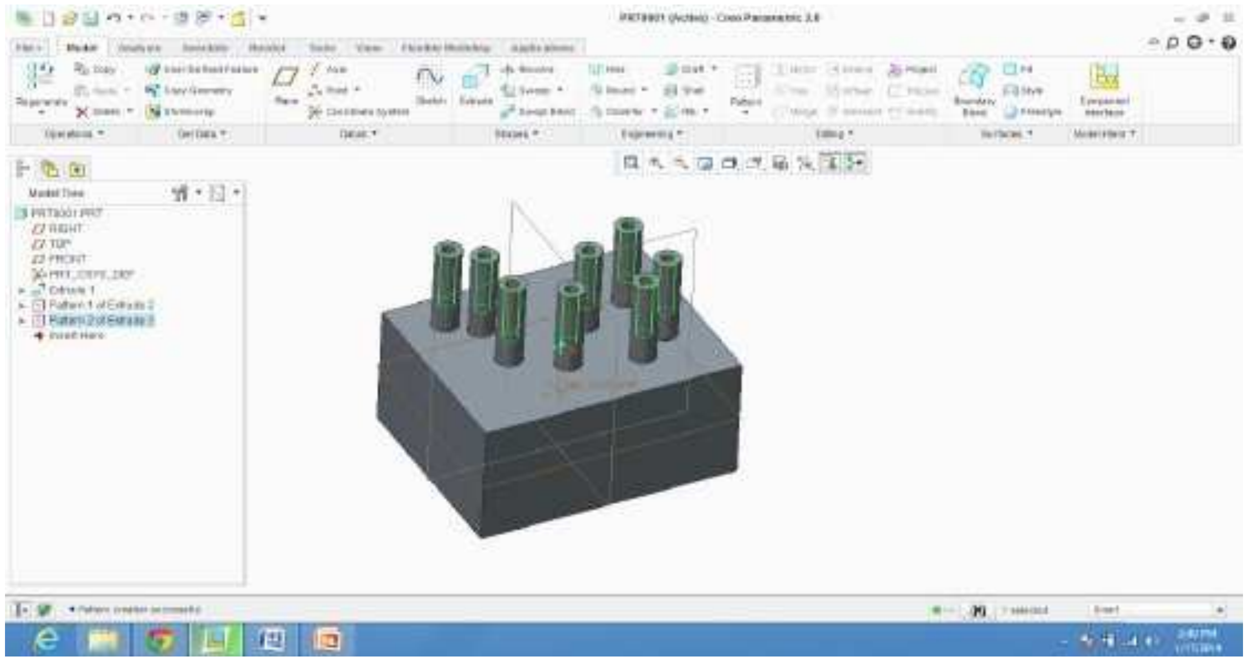
Select feature> goto pattern> use geometry points to create points>manage location of point > click on done



7> REFERENCE PATTERN

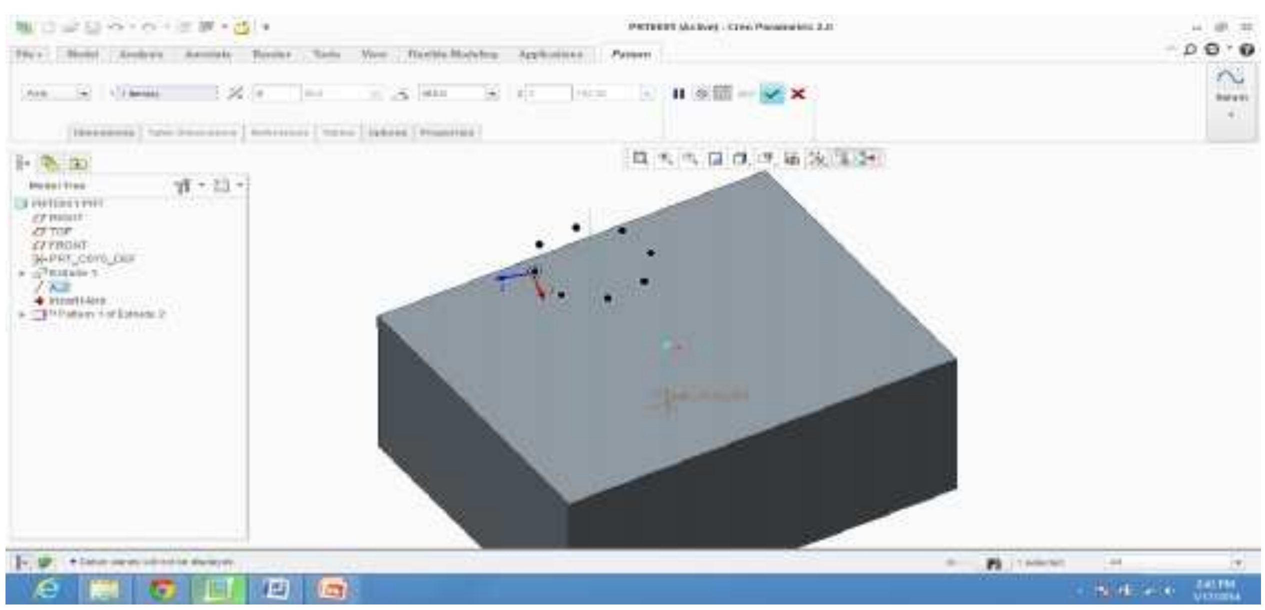
A pattern that references already created pattern is known as reference pattern.

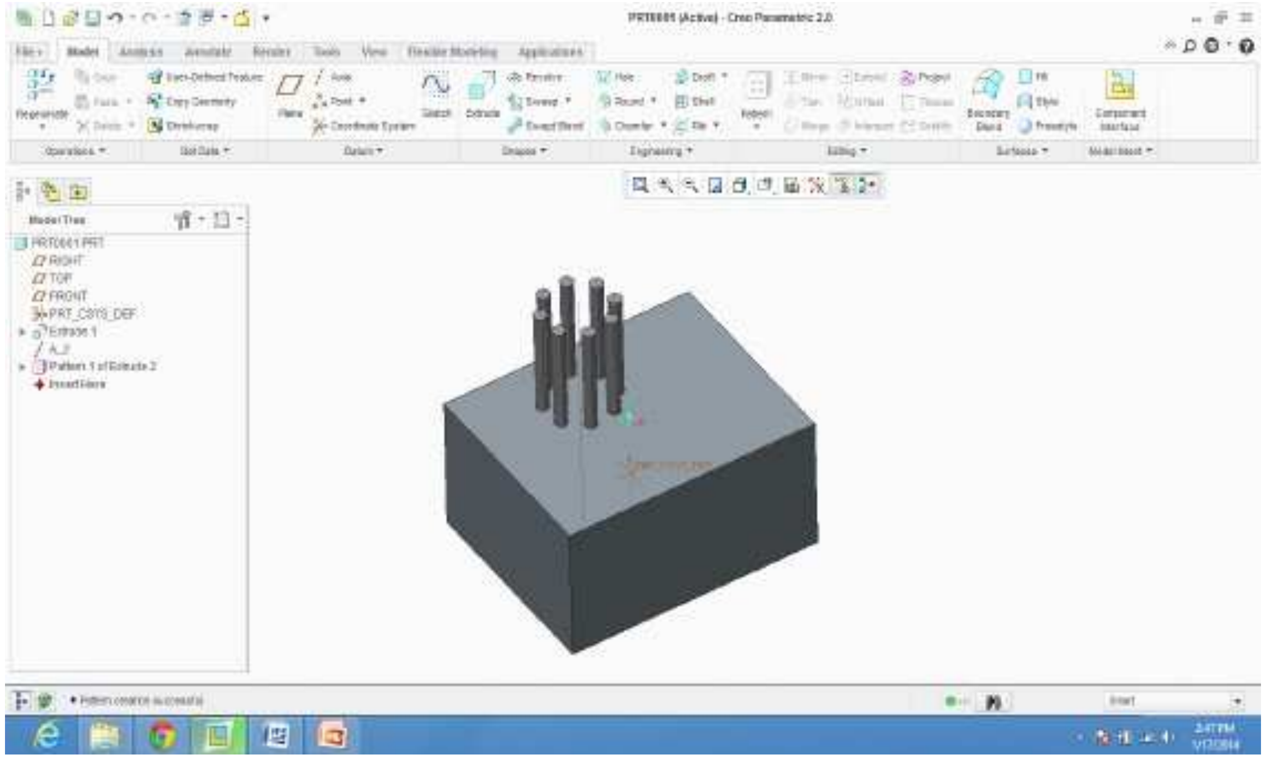




8> AXIS PATTERN

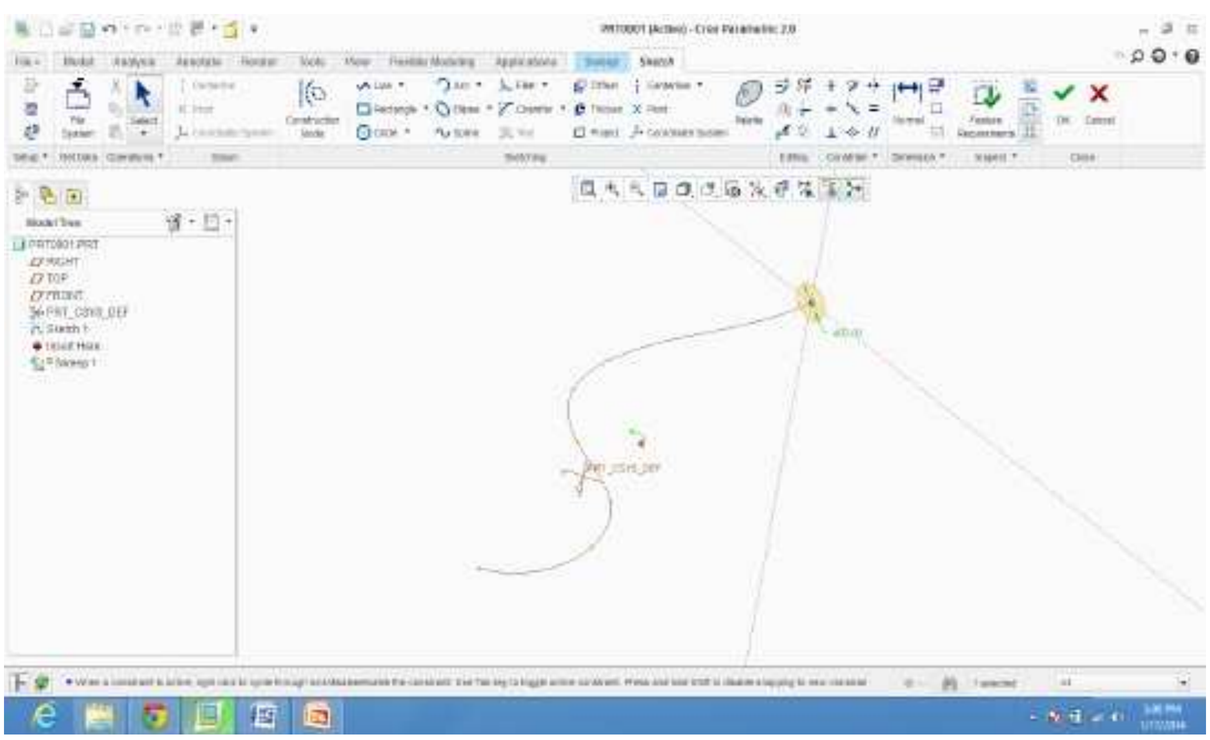
Select feature >goto pattern> select axis> define angular extent> total number of copies> click on done

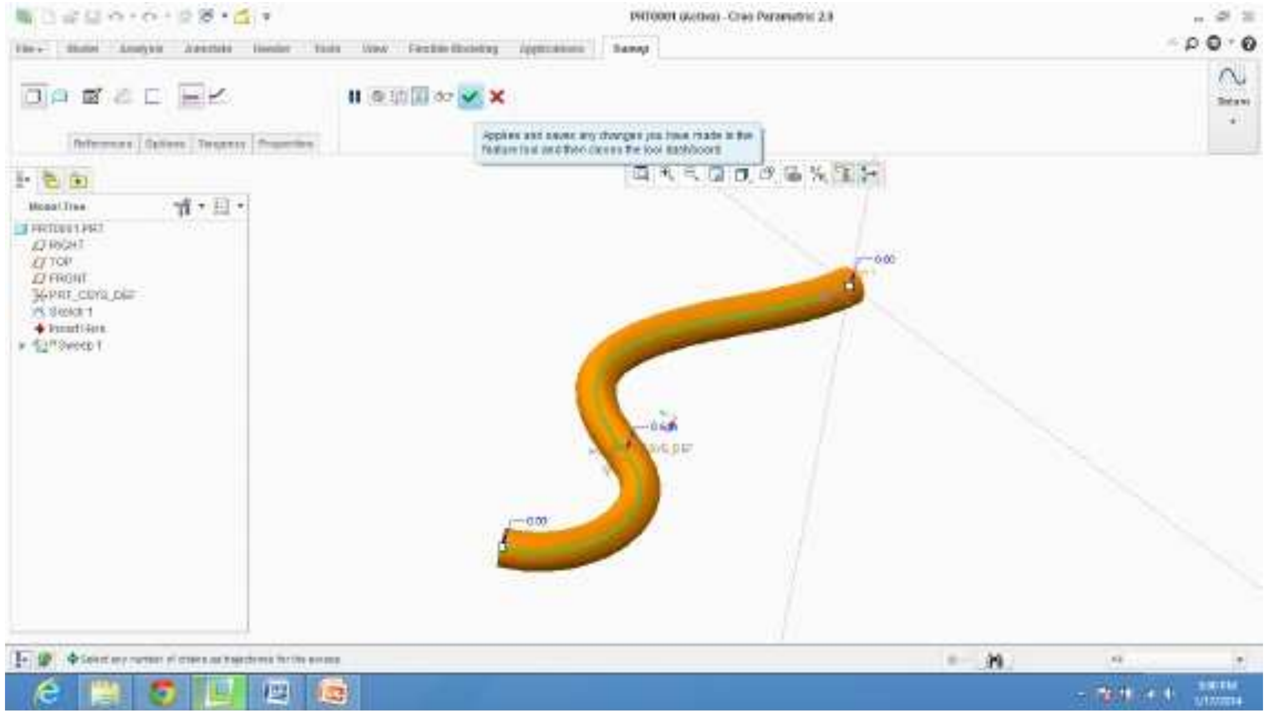




CHAPTER 8
CREATING SWEEP FEATURES

Sweep is an addition or removal of material along a trajectory.
Application: chasis design, pipeline , columns , beams



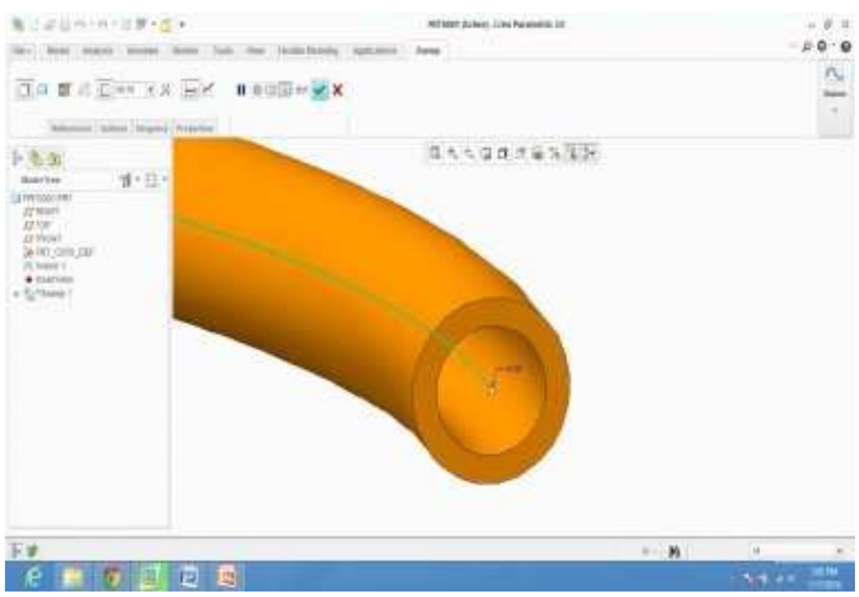


Step 1> Create the trajectory in sketcher module

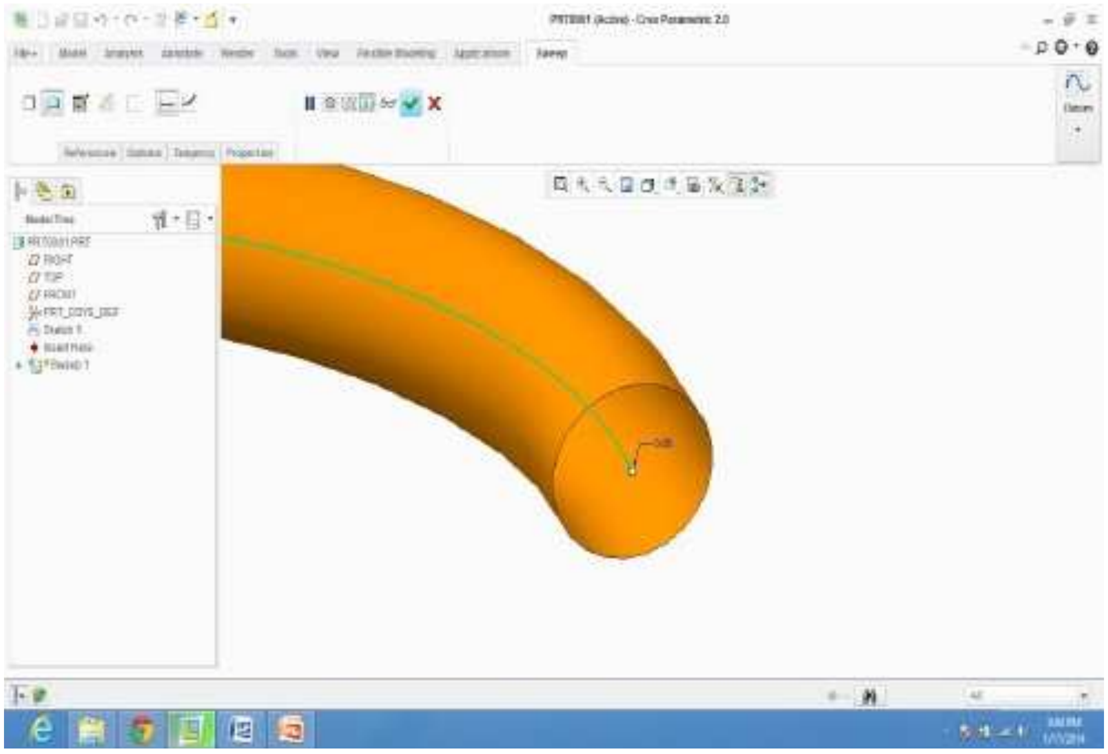
Step 2> Goto sweep>references>select trajectory > goto create/edit sweep section > create the sweep section

Note : To sweep as a solid section must be closed . Trajectory may be open or close .

THICKEN SECTION



SWEEP AS A SURFACE

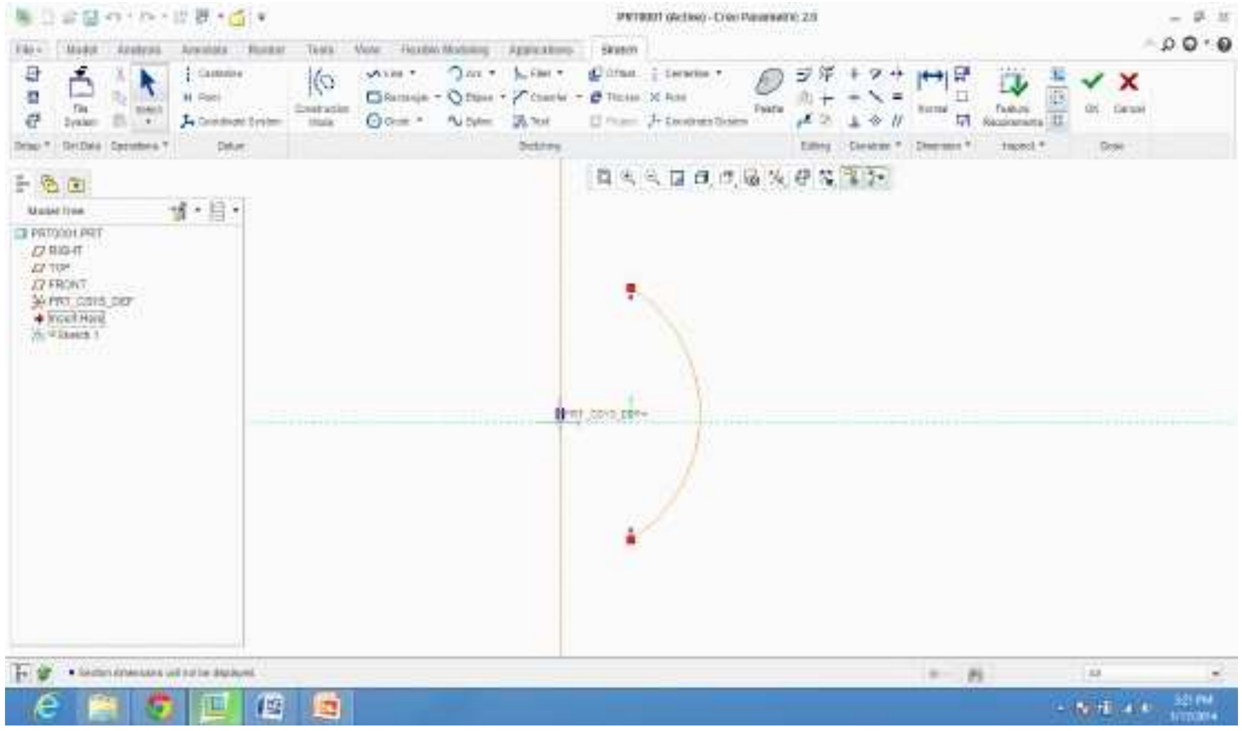


CHAPTER 9
CREATING HELICAL SWEEP FEATURES

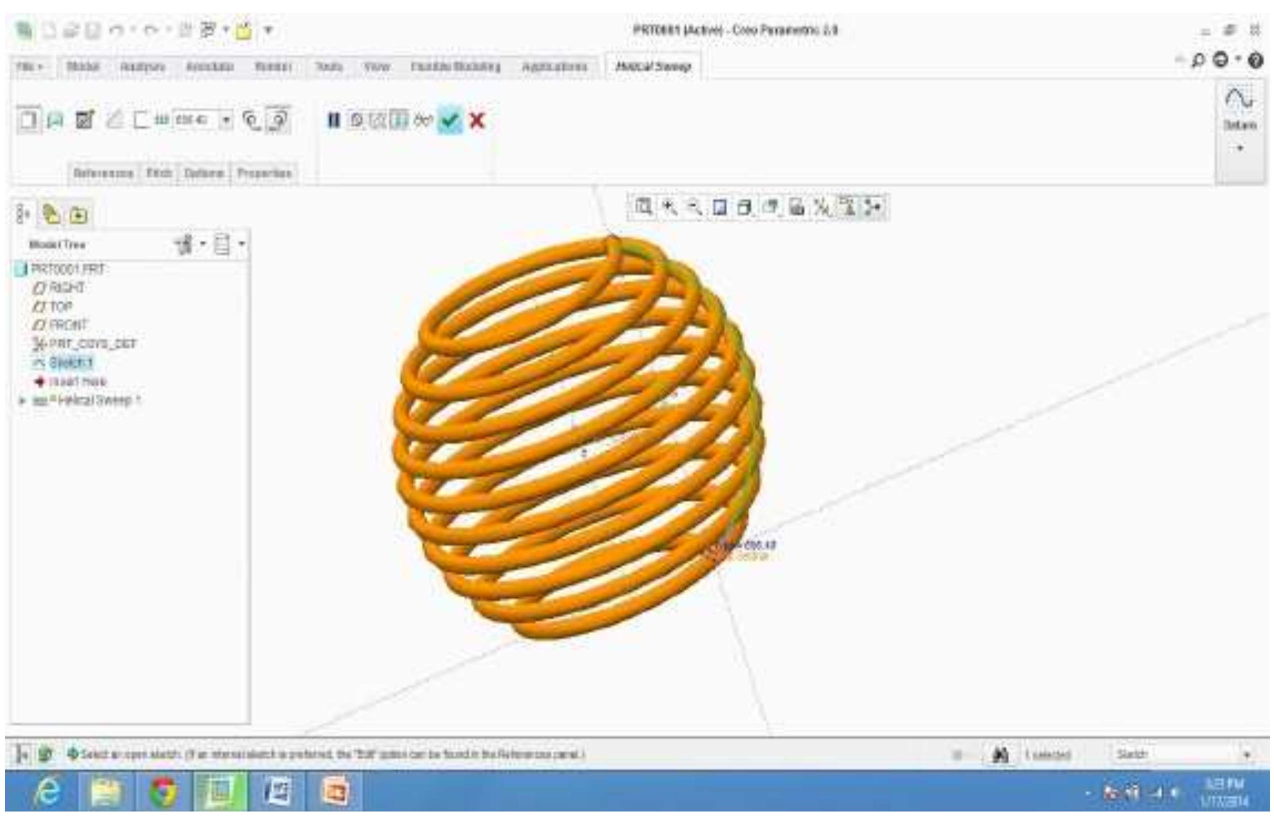
Step 1: Goto helical sweep > references > select plane> create axis of revolution > create sweep profile> create cross section > change pitch value or flip > done

Note : You can create left handed or right handed threading also using this tool.

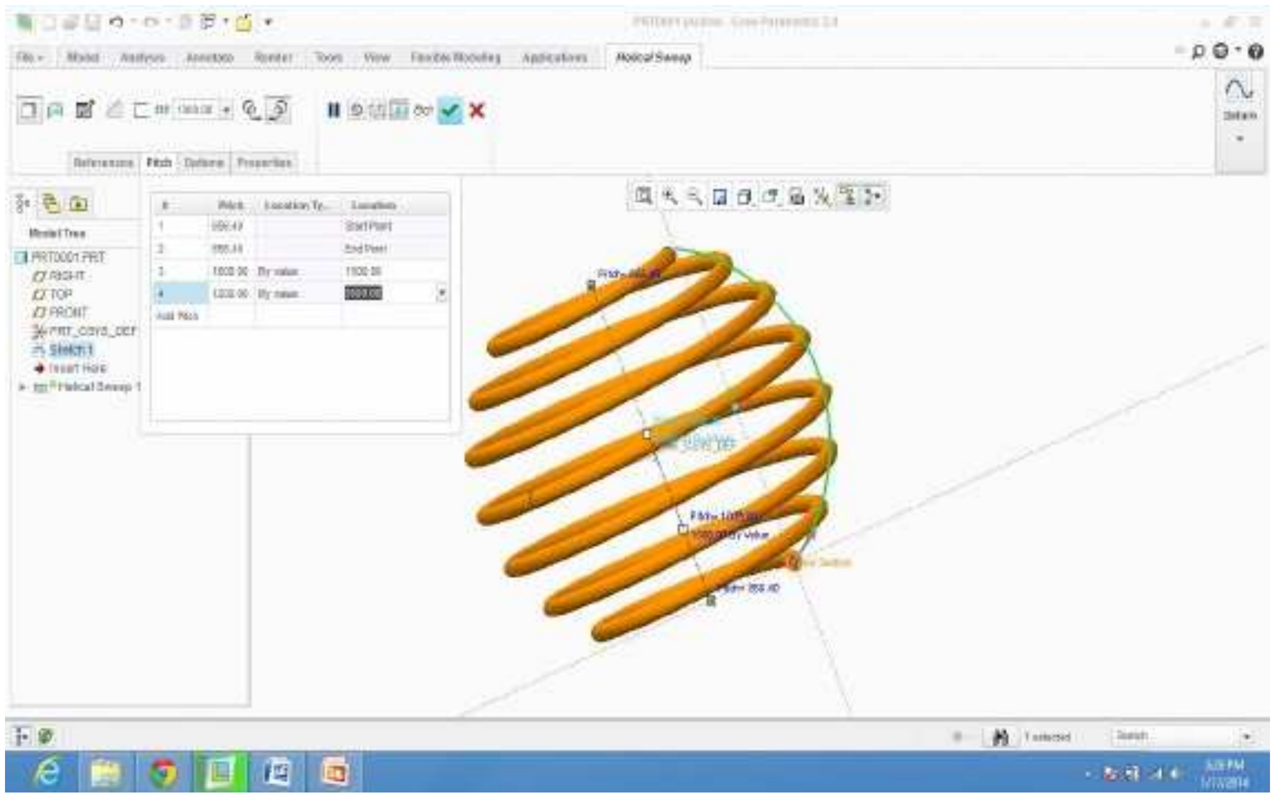
Application: Suspensions,threading



FOR CONSTANT PITCH VALUE



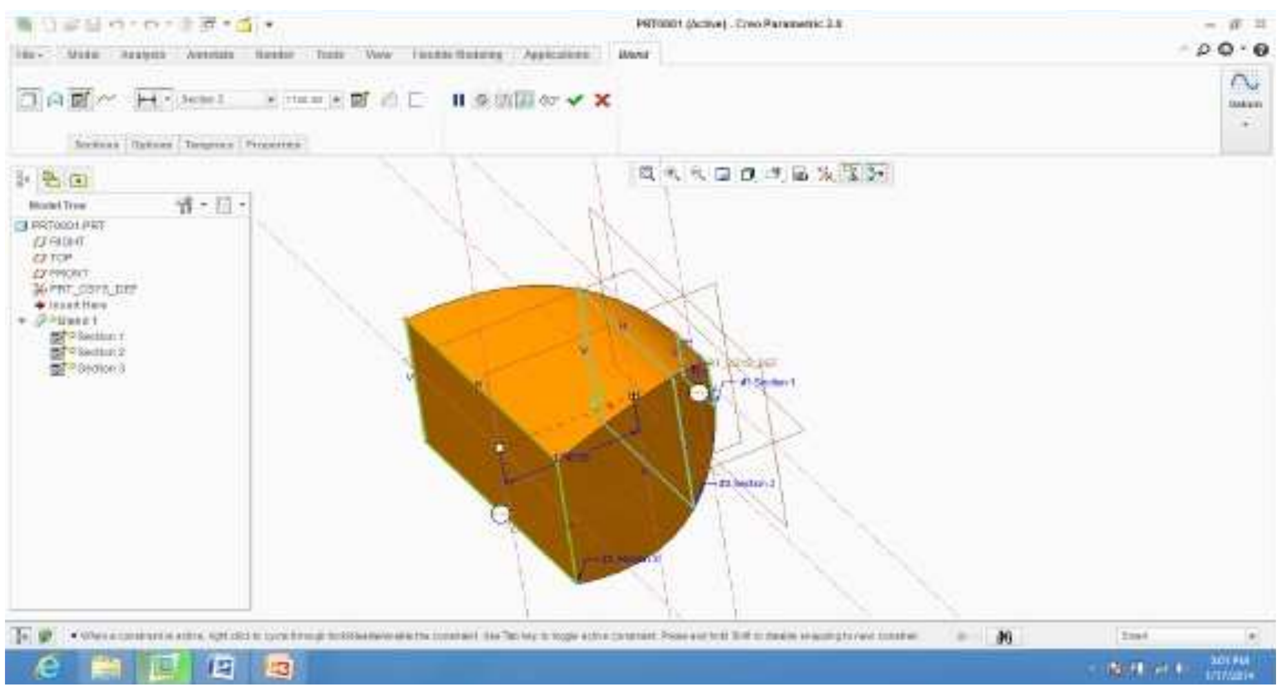
FOR VARIABLE PITCH VALUE



CHAPTER 10
CREATING BLENDS

Addition or Removal of material along straight path consisting of different sections.

Steps :Go to shapes > blend> sections > define> select the plane > sketch > sketch view> create 1st section > done> specify distance between sections > goto sections > sketch > create 2nd section



Options :

1> Straight : In straight vertices are connected by lines

2> Smooth: In smooth vertices are connected by splines

Note:

1> The number of entities per section must be equal

2> Line connects from start point to start point

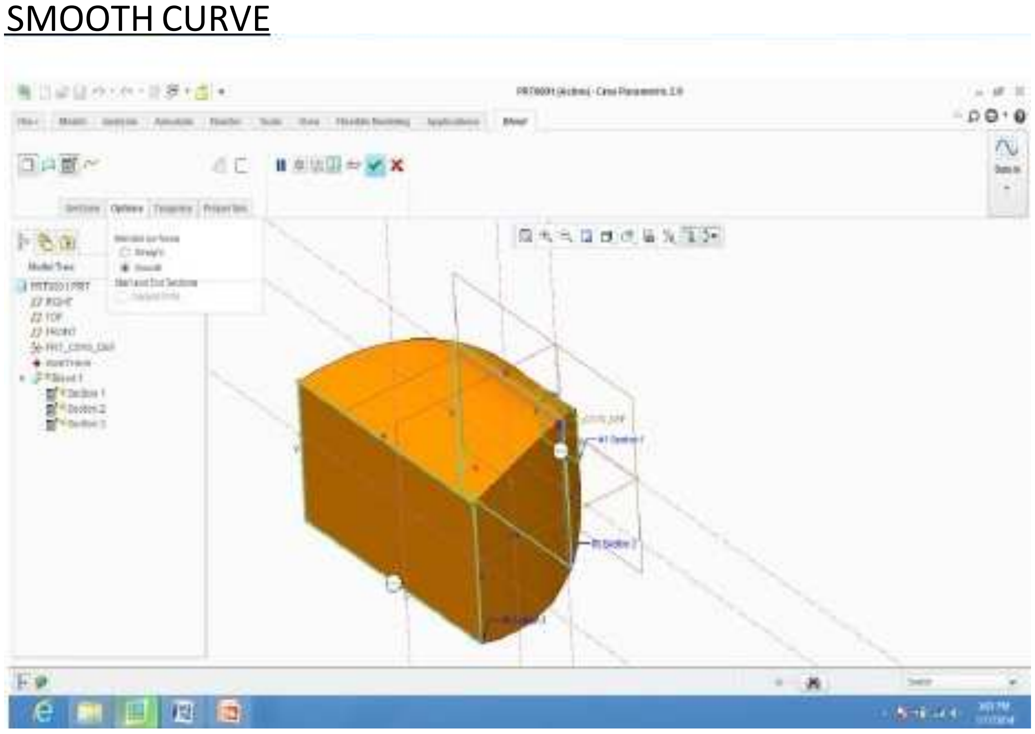
3> You can make same cross section at different and can rotate and resize in order to twist the cross section .

4> To make a point as start point left click on point>right click > start point

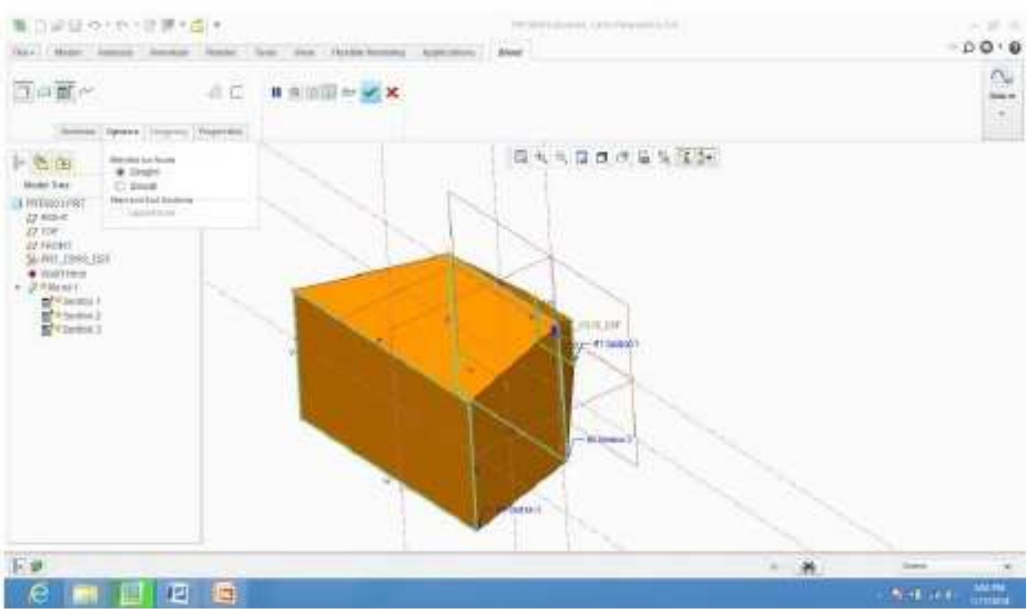
5> Minimum 2 sections are required to create blend feature.

Divide : This tool is used to divide single entities into multiple entities.

<u>SMOOTH CURVE</u>



<u>STRAIGHT CURVE</u>



CHAPTER 11
CREATING SWEPT BLENDS, COPY AND PASTE FUNCTIONALITY

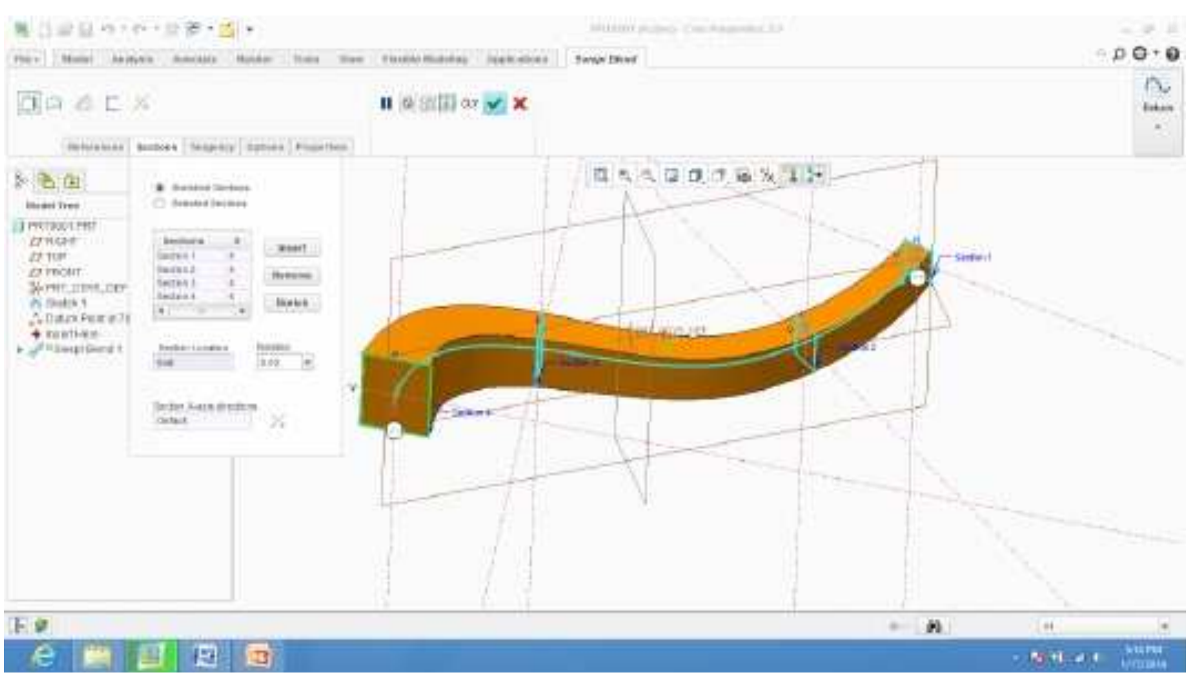
This is a combination of sweep and blend.

Steps : Goto swept blend>select curve>goto sections>create 1^{st} section at start point >goto insert>select 1^{st} point>goto sketch > create 2^{nd} section>goto insert > select 2^{nd} point> create 3^{rd} cross section > repeat above steps ….> create last section at end point (you can use construction point to end the section)

Note:

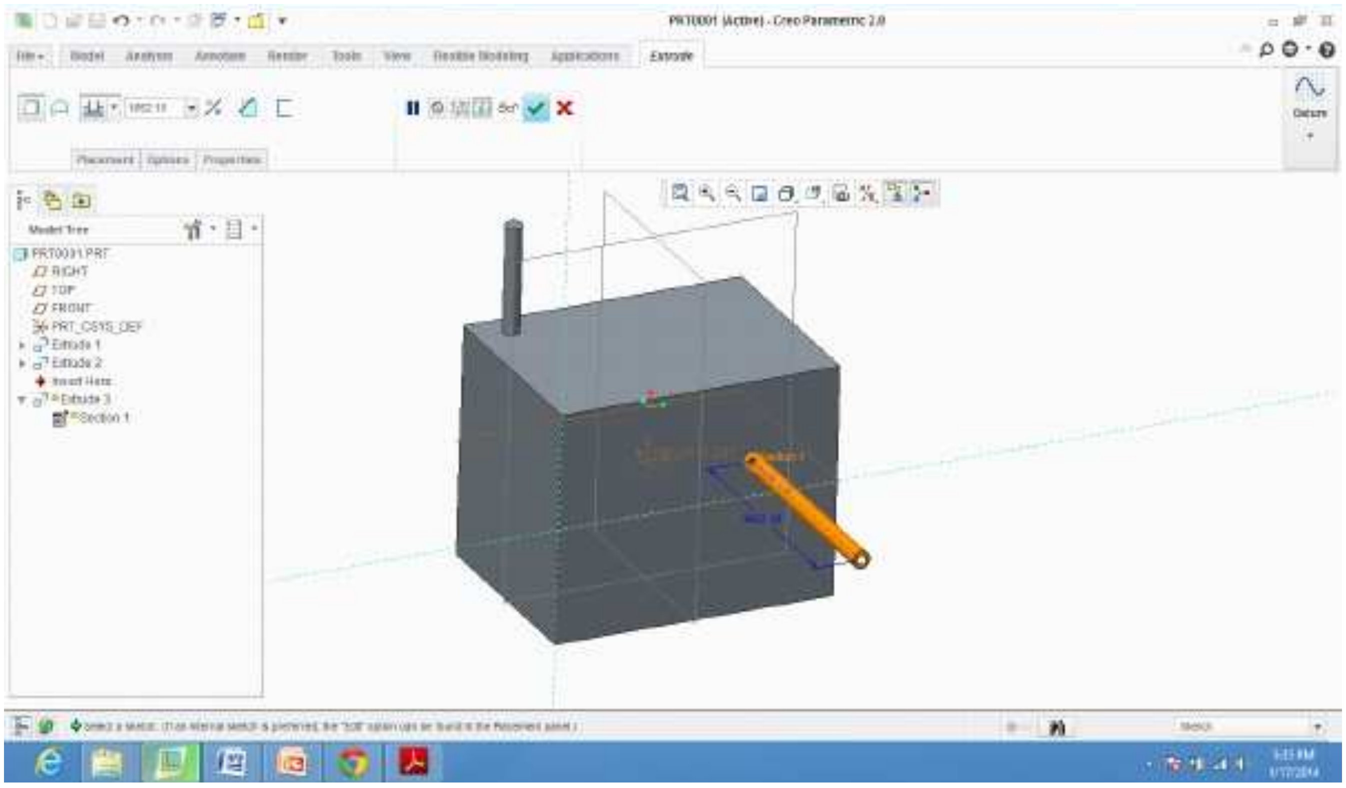
1> The number of entities per section must be equal
2> Line connects from start point to start point
3> You can make same cross section at different and can rotate and resize in order to twist the cross section .
4> To make a point as start point left click on point>right click > start point
5> Start point and end points are default provided.

Application : Silencer design , crane hook , air plane body design .

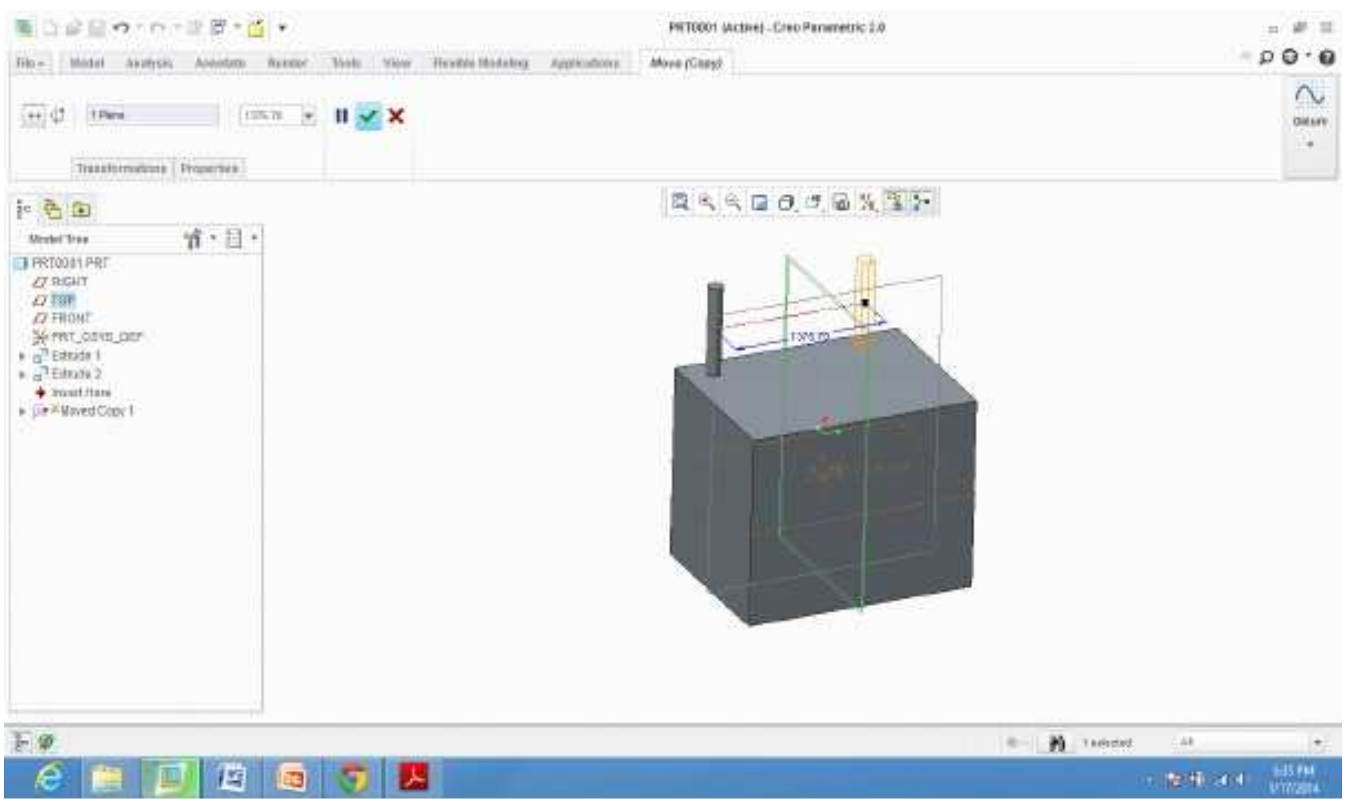


COPY AND PASTE FUNCTIONALITY

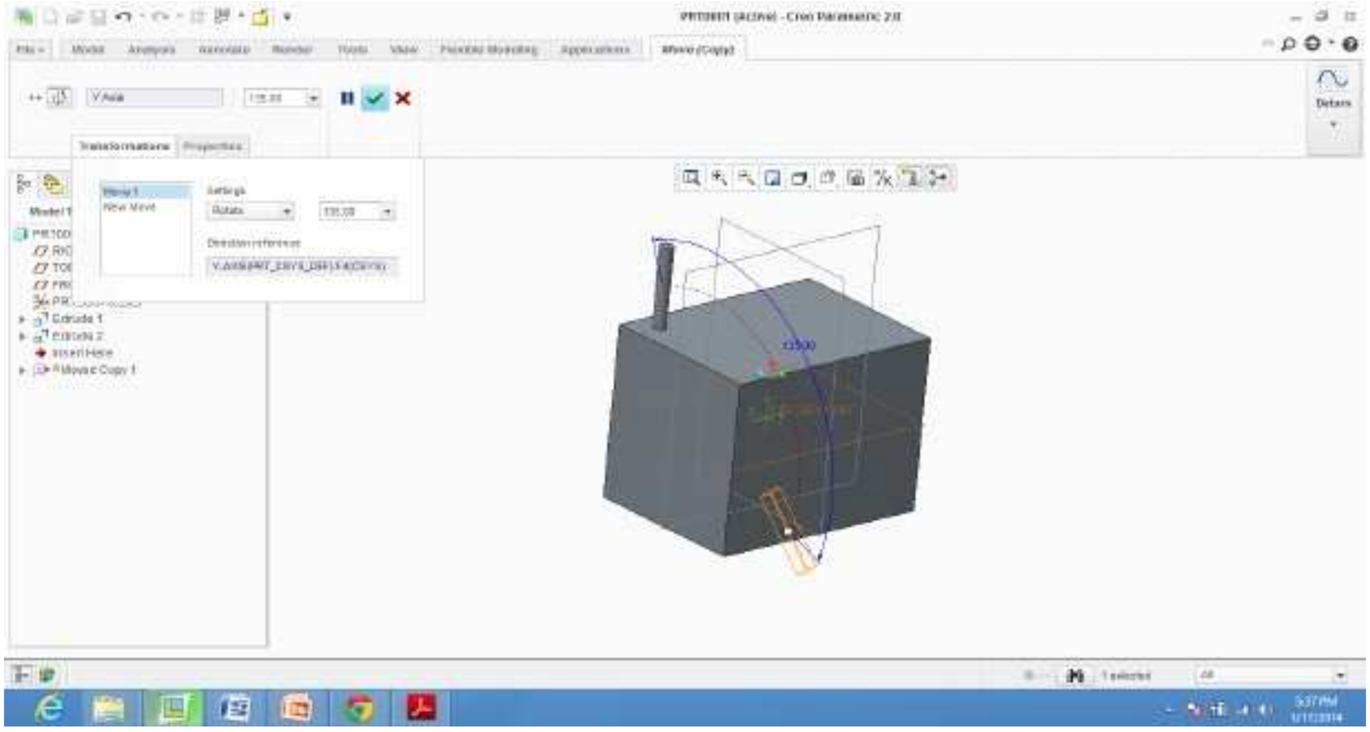
The copy and paste functionality enables you to quickly duplicate a feature .



You can use paste special option to apply move and rotate options to the resulting copied feature



ROTATING COMPONENTS



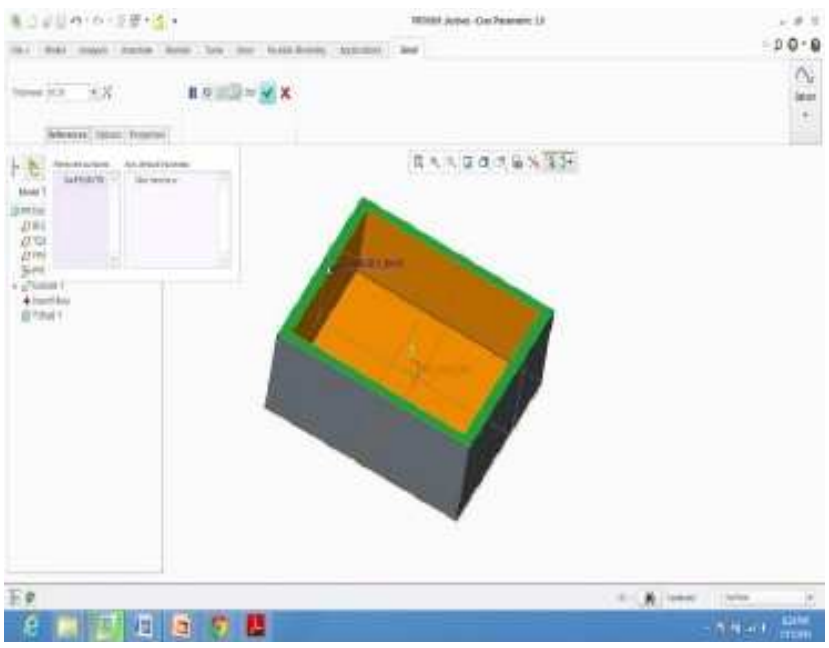
CHAPTER 12
CREATING SHELLS, DRAFT, HOLES, RIBS

SHELL : The shell feature hollows out the inside of a solid model , leaving a shell of a specified wall thickness .

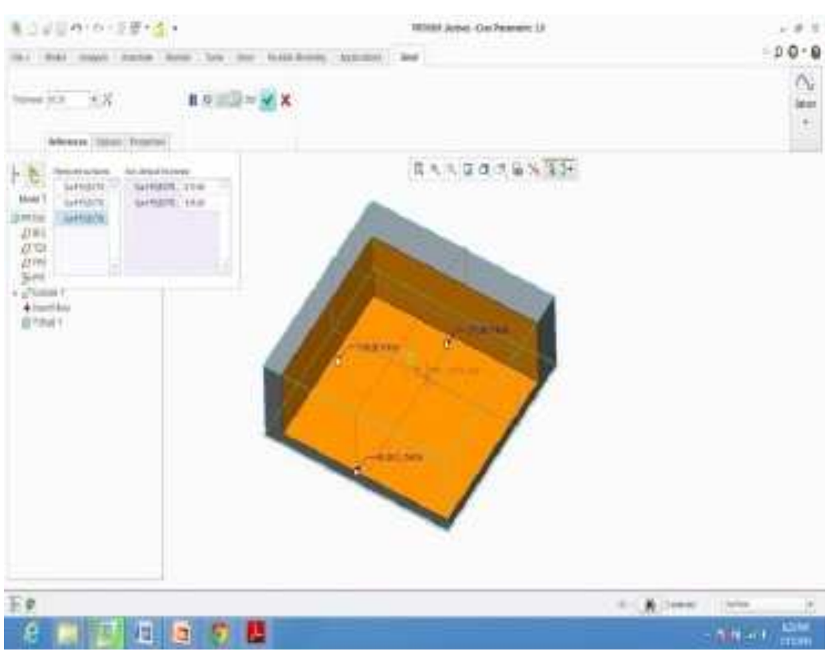
Creates Shell feature

i) Select planer surface(for more faces select with Ctrl)

ii) Specify Shell thickness



For different thickness and multiple openings use ctrl and non default thickness option

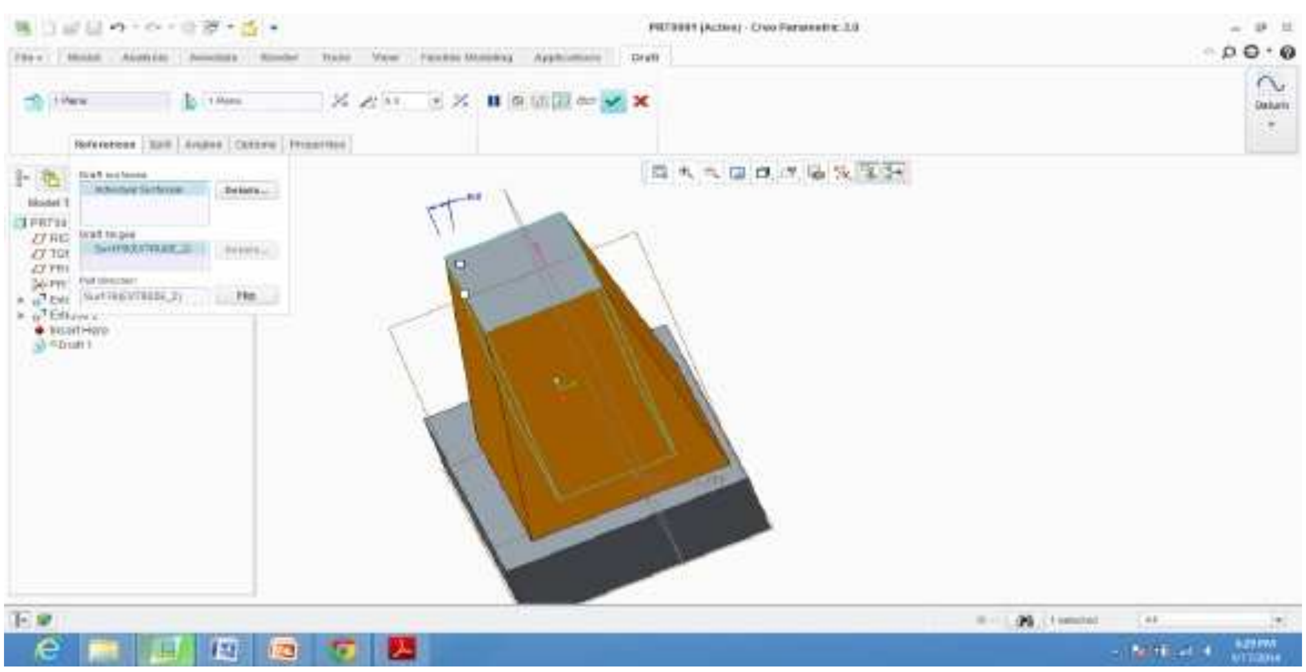


DRAFTS : Draft features are typically used as finishing features in molded and cast parts.

Draft features consist of:

1> Draft surfaces
2> Draft hinges
3> Pull direction
4> Draft angles

- Go to/reference tab/select draft surface then click the surface you want/select the draft hinges then click the surface for hinge/select the pull direction.
- Right click on the white circle of draft angle to create more draft angles.
- 3 Degree is the maximum draft angle in Industries. Software permits +30 degree to -30 degree as draft angle.



HOLES

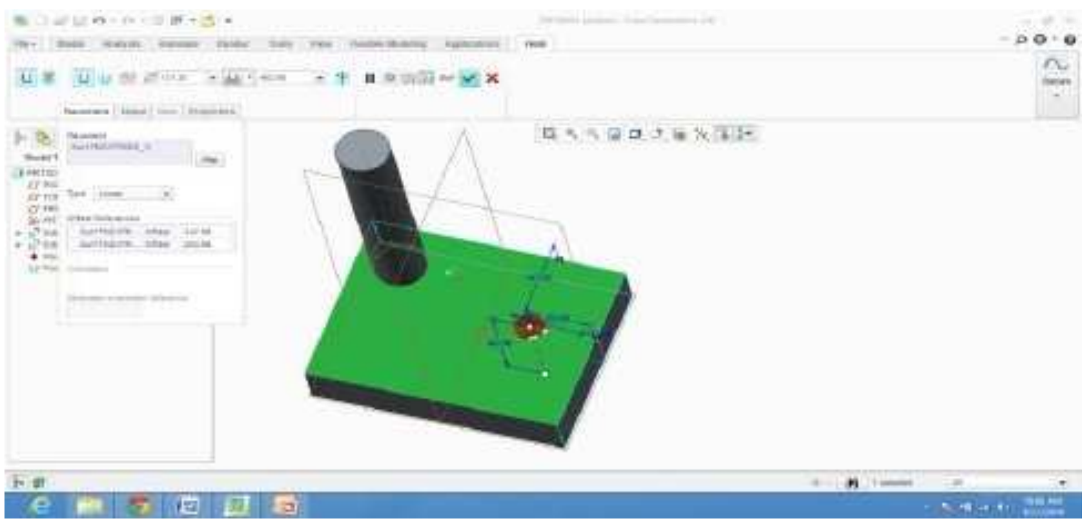
Four methods to create Holes.

- Linear Holes

A> Go to/Holes in dress up feature/select surface where you want to create a hole.

B> Specify primary and secondary references

Specify the references with the edges of feature

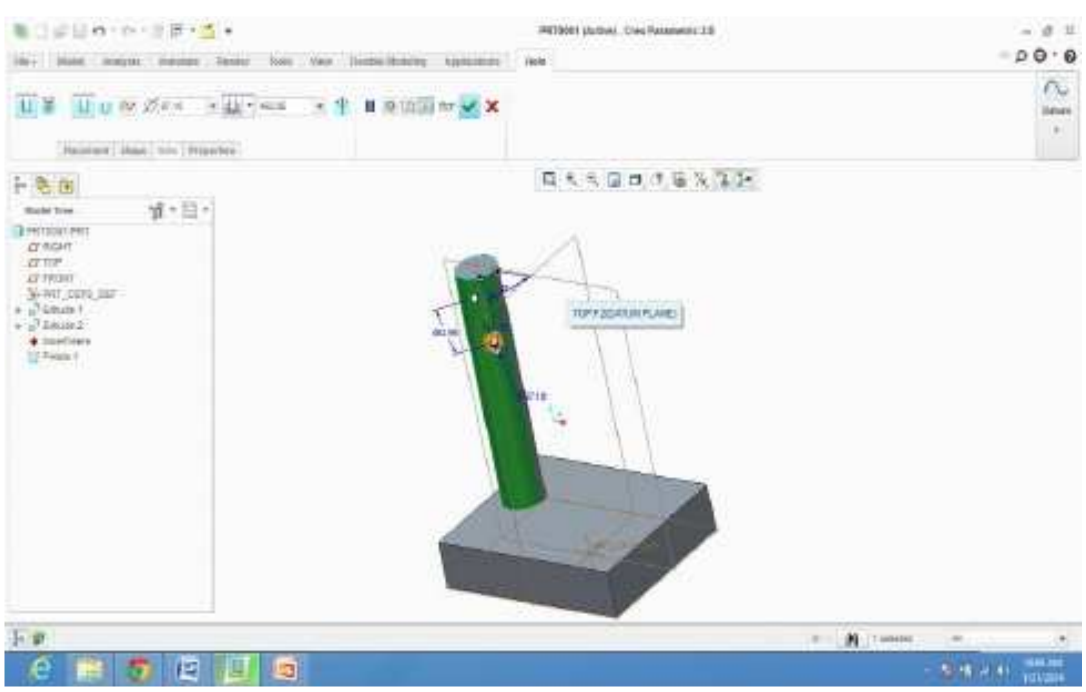


- Radial Holes

A> Go to/ Holes/select side surface of cylinder.

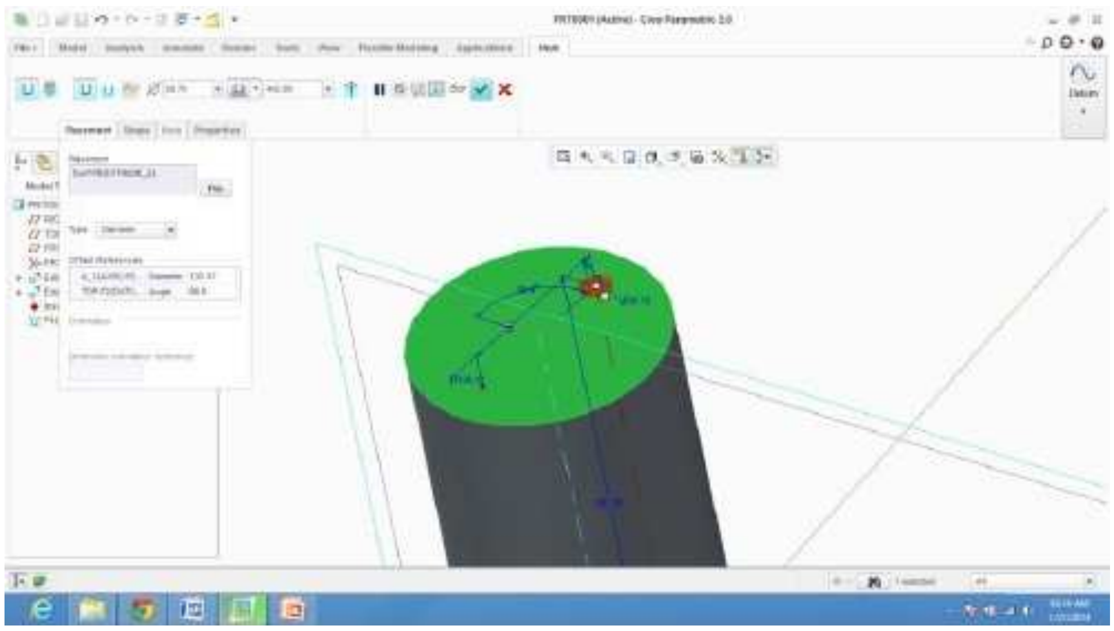
B> Specify primary and secondary references.

C> 1^{st} with Top surface and 2^{nd} with plane(For Angle).



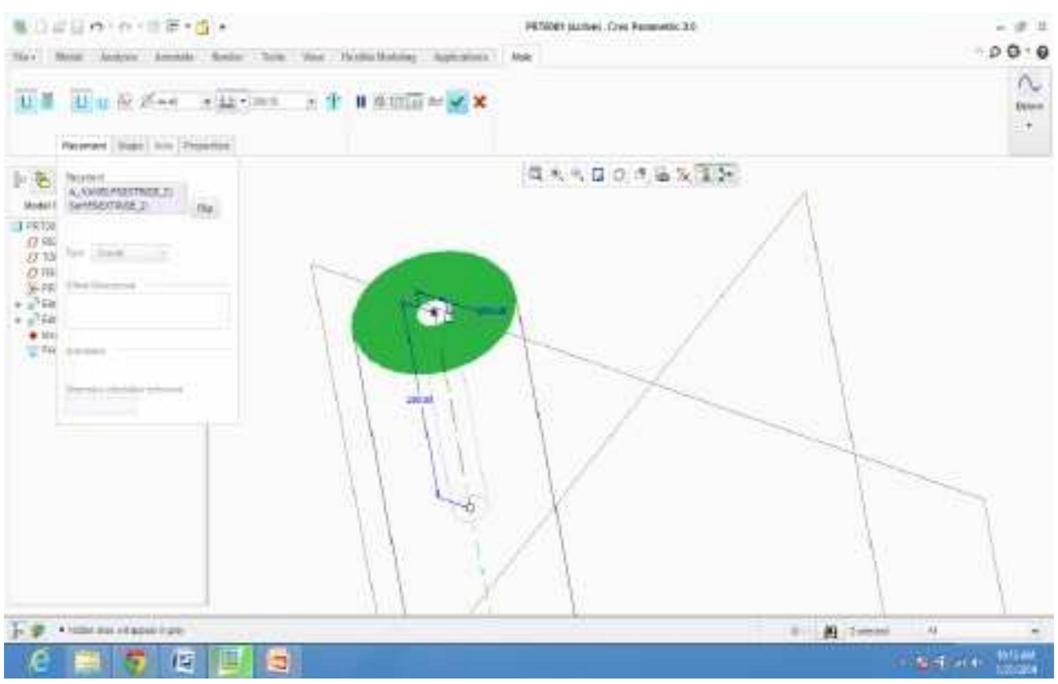
- Diametric Holes

A> Select the top surface of cylinder.
B> Specify primary and secondary references.
C> Specify 1st with axis and 2nd with plane.
D> Manage angle and diameter.



- Coaxial holes

A> Select axis of cylinder for co-axial hole.
B> Select top surface with holes.

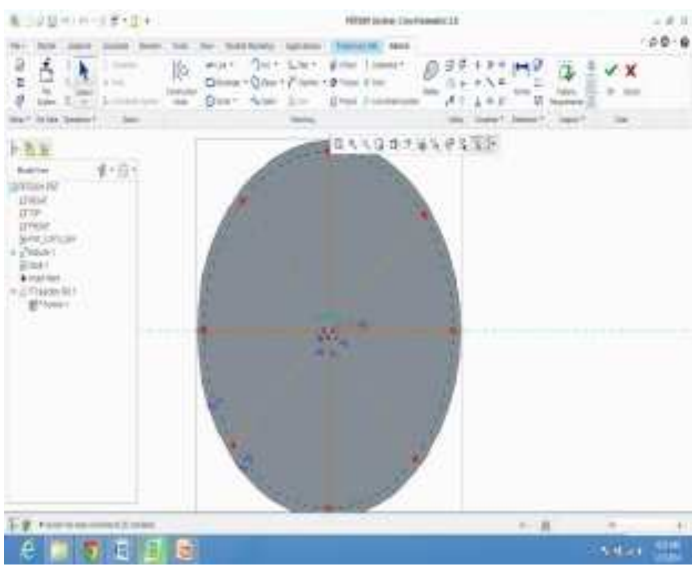


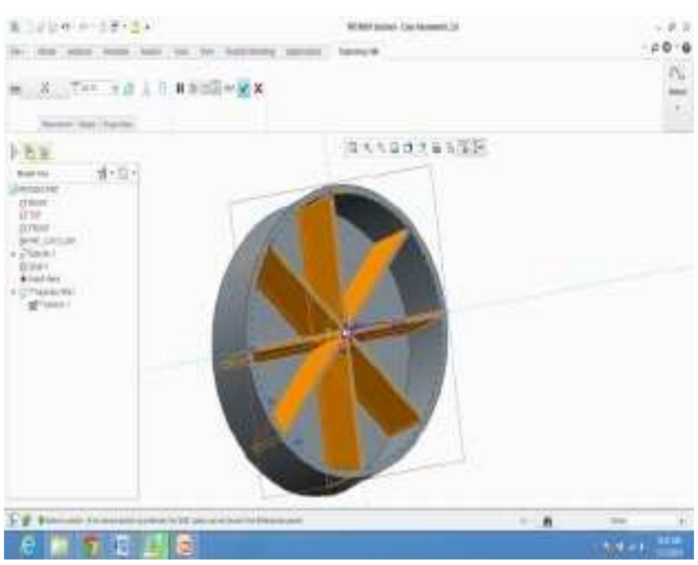
STANDARD HOLES

A> To create standard holes click on the icon(standard hole) above placement tab
B> Standard holes from M1 to M68 are available in CREO.
C> You can create ***counter sink*** and ***counter bore*** typed holes.
D> Thread will be shown in violet color.

TRAJECTORY RIB

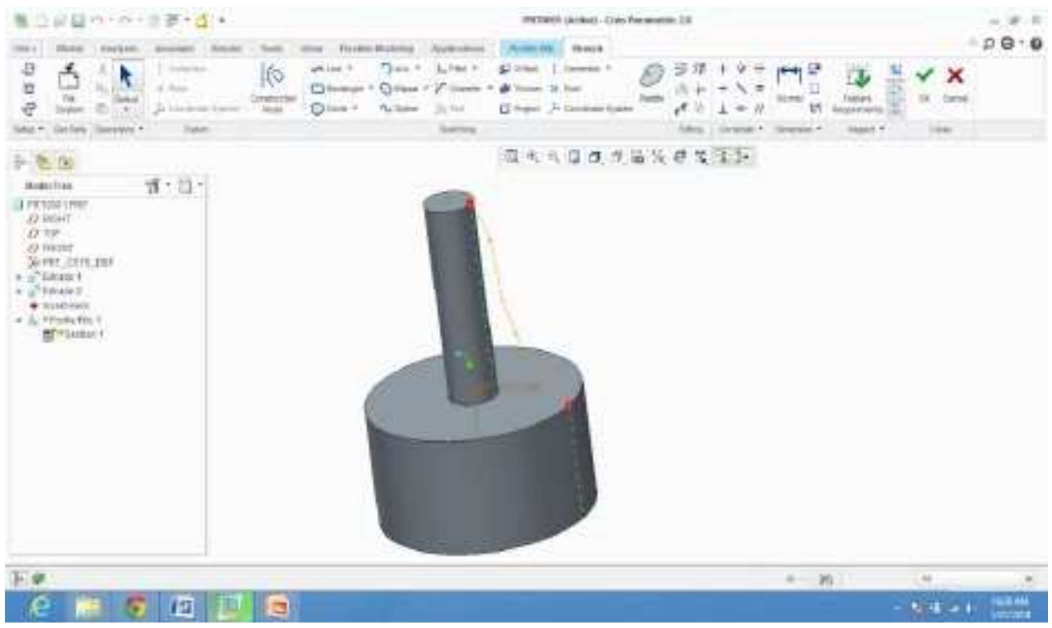
Goto Rib> Trajectory Rib> Select top surface> create top surface> create section > add material upto next surface

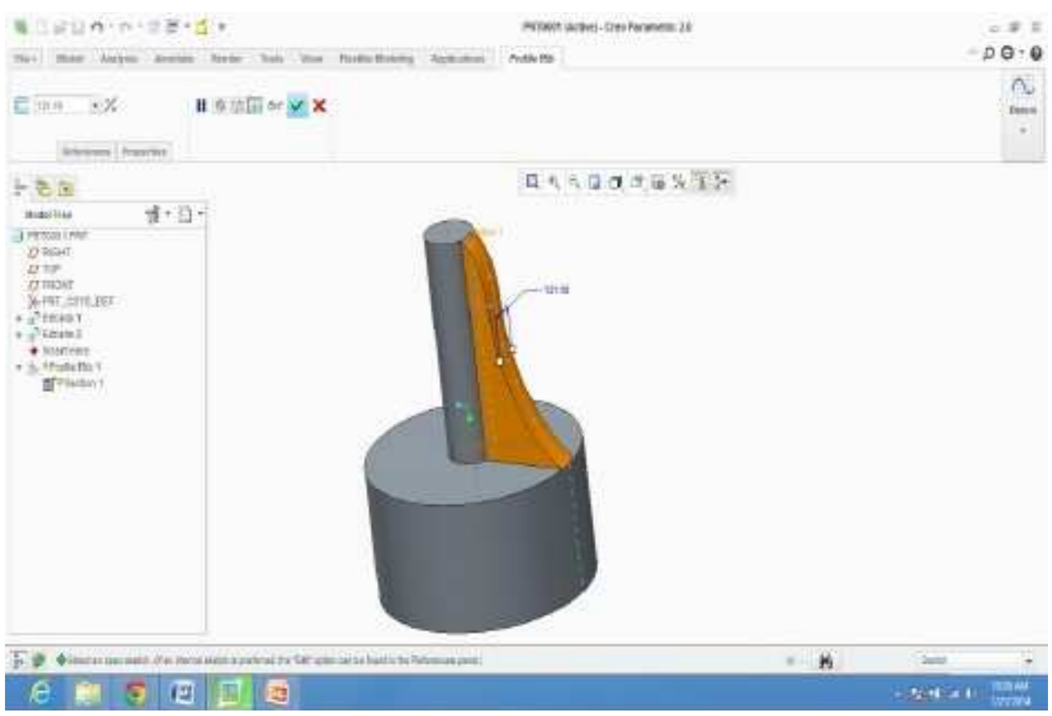




PROFILE RIB

Goto profile rib> reference>define plane> create sketch > done





CHAPTER 13
RELATIONS AND PARAMETERS, DELETING, SUPPRESSING

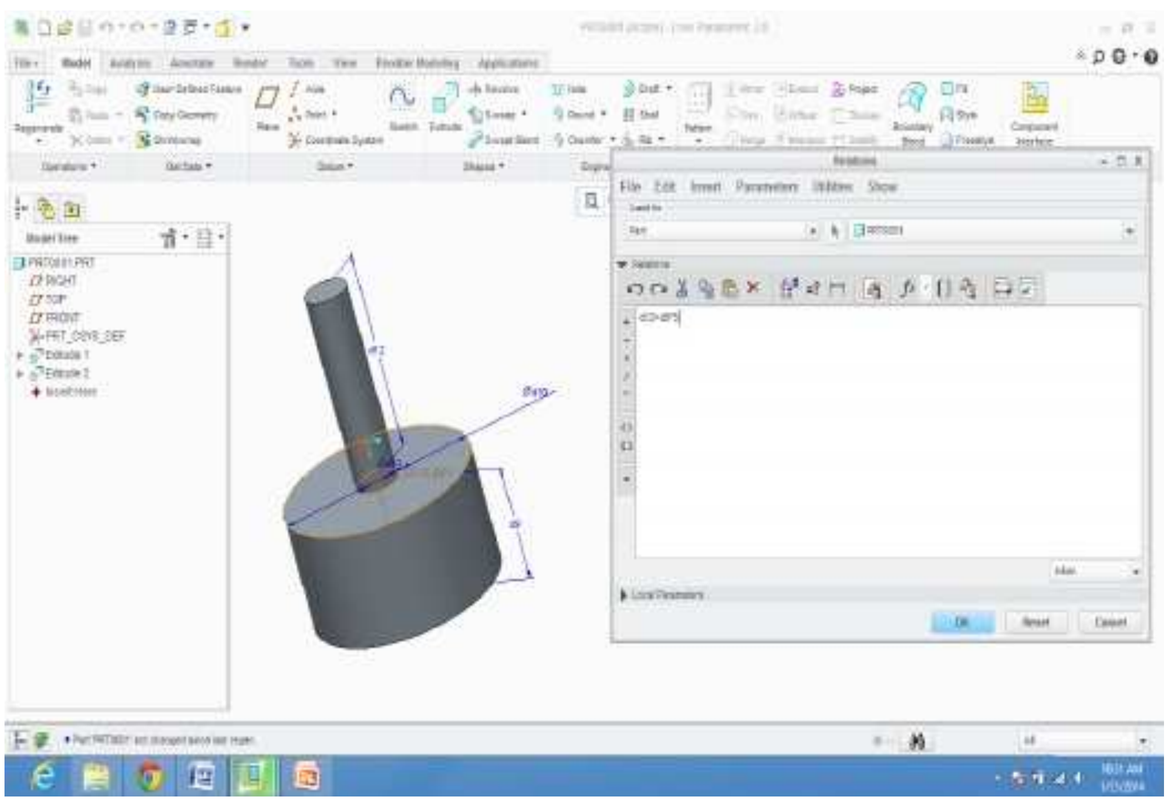
Relations are user defined equations written between dimensions and/or parameters . Relations let you express design intent by defining relationships within sketches , features , parts or assembly components .

Relation Uses :

1> To define values for dimensions based on other dimensions .

2> To set maximum or minimum constraints for dimension values .

3> To describe conditional relationships (ENDIF , ELSE) between dimensions



Parameters are meta data information that can be included in an assembly template or created by a user in his own part or assembly .

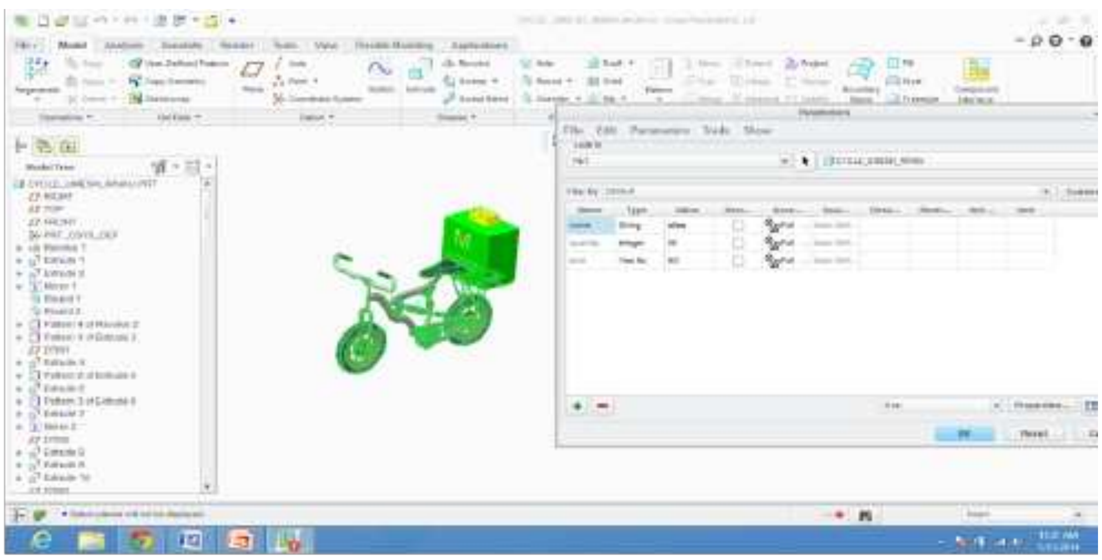
Types of values accepted by parameters

1.Real numbers : example 25.55 , 1.66667

2. Integers : example 1 , 5, 250

3. String : Any consecutive sequence of alphanumeric characters

4.Yes/No:



Suppressing and deleting

> Suppressing an item removes it from the graphics window and regeneration cycle , but the item can be resumed .

How to suppress ?

Rt click on feature > suppress

> Deleting is permanent .

Rt click on feature > Delete

Both suppressing and deleting items follows parent child relationship. Suppressing items causes regeneration speed to increase . Suppressed item is denoted by black square .

CHAPTER 14
MANAGING DESIGN INTENT, ANALYSING MASS PROPERTIES

There are three methods to manage design intent :

1> Reordering : You can reorder features in the model tree by dragging them to a new location

2> Inserting: You can insert new features in the model tree wherever desired

3> Redefining: You can redefine features in the model tree by
Selecting feature > right click > edit definition

You can redefine in following areas
Feature Type
Size
Shape
Location
Options
References

While designing any cad model we may need to analyze its mass properties viz . including mass, weight, centre of gravity, moment of inertia, product of inertia.

Step 1: Goto file> prepare> model properties (a window will pop up) > goto material> change>assign material > click on ok.

Step 2: Goto analysis tab> mass properties(mass property window will pop up) > click on coordinate system > click on 'I' i.e. info icon

CHAPTER 14
SURFACE MODELLING TOOLS

- It is a method of modeling with zero thickness of boundary.
- uses of surface modeling

  E.g. (car bonnets, bikes fuel tank, aerofoil etc.)
- SURFACE : Surfaces are infinitely thin, non –solid features used to aid in the design of highly complex and irregular shapes.
- Surfaces are shown using orange and purple highlighting on the edges

  - Orange denotes open or outer edges.

  - Purple denotes closed or two-sided edges, since they border two surface patches

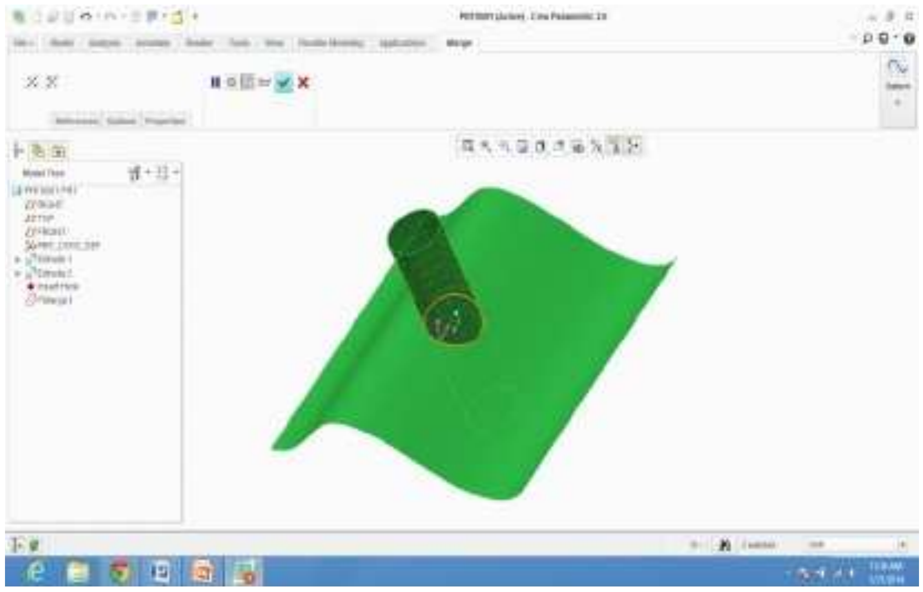
- QUILTS : A quilt may consist of a single surface or a collection of surfaces. A quilt represents a patchwork of connected surfaces. A multi-surface quilt contains information describing the geometry of all the surfaces that compose it, and information on how these surface are joined or intersected.
- Surface Patch : If you create a surface feature, which is made of several segments, the surface is created with multiple patches.

Tools used in creo

- ***MERGE***

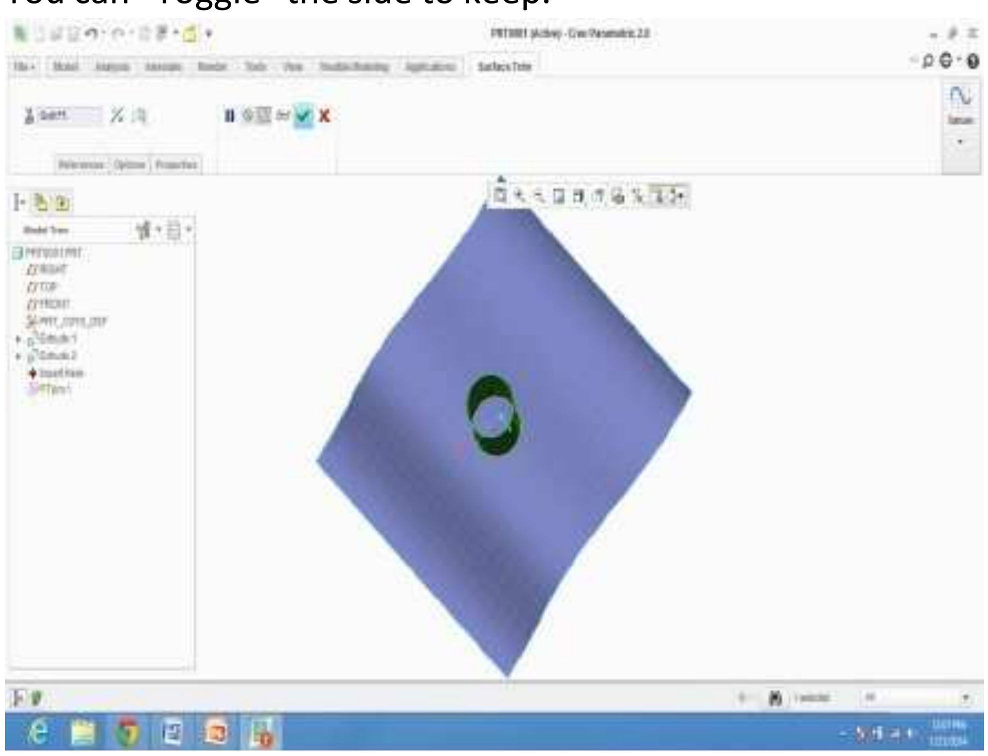
  In Pro-E surface modeling, most of the time you will build a surface model by creating pieces of surfaces adjacent to the other. In order to convert the surface model to a solid part you have to first merge all the surfaces. For this purpose you have to use the **"Merge"** option.

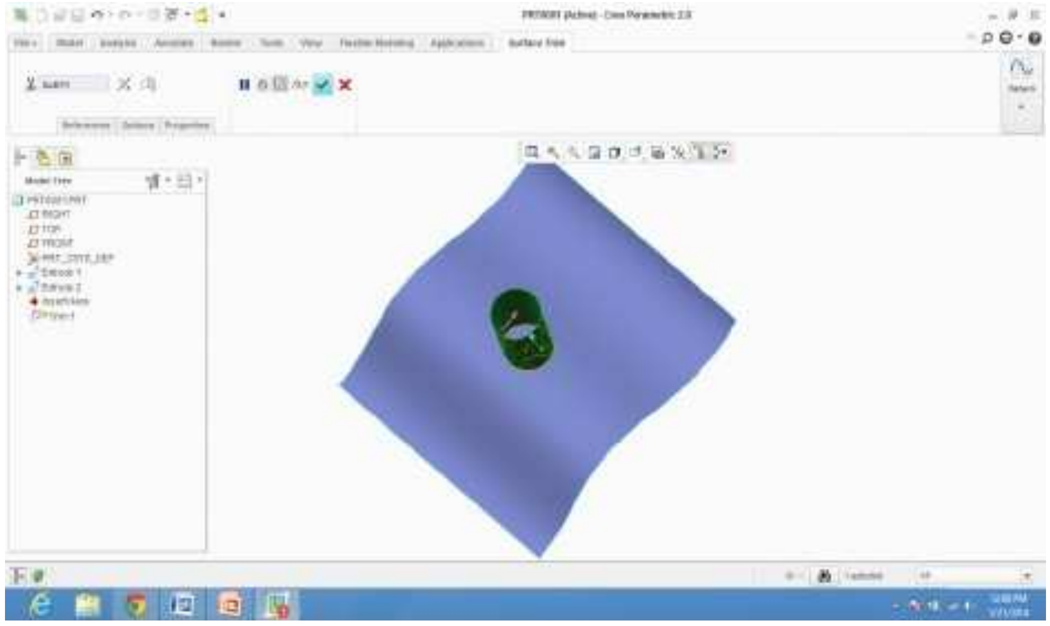
  Click the "surface-1" and "surface-2" by pressing **Ctrl** button and then go to ***Merge***. The Merge dashboard dialogue box will appear where you can change the directions for each of the surfaces



⦿ *TRIM*

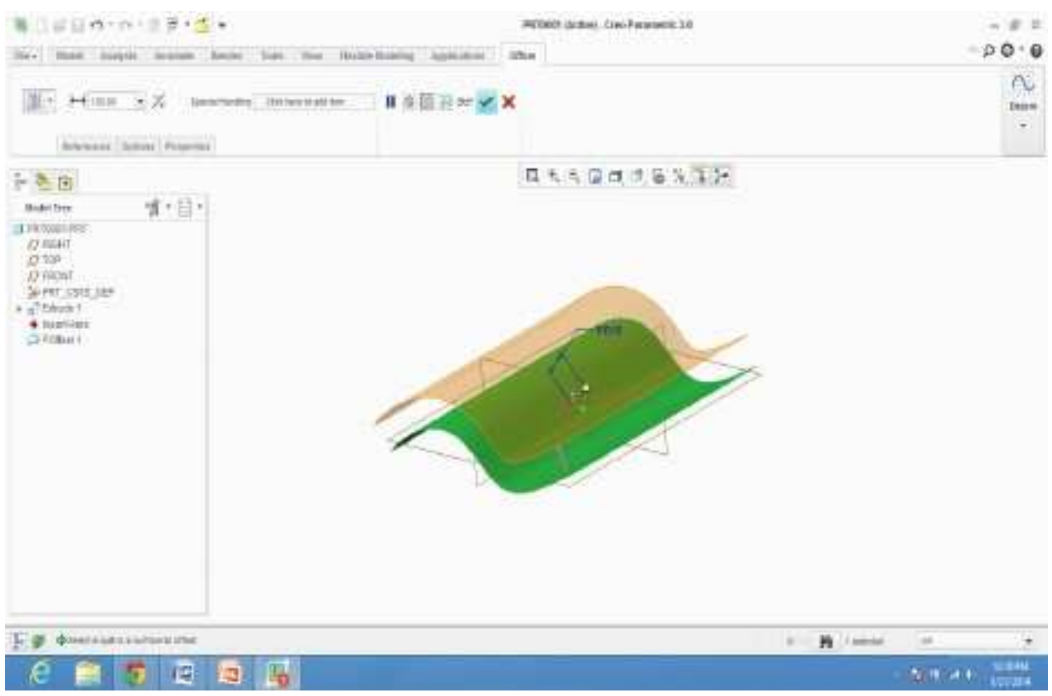
For trimming a surface by another surface curve, the two entities need to be intersected with each other. Say, we need to trim the “Surface-1” with the “Surface-2” of the Figure-1. Click the “Surface-1” and go to ***Trim,*** you will get the “trim dashboard” appear; now you need to select the “Surface-2” as the trimming surface. You can “Toggle” the side to keep.

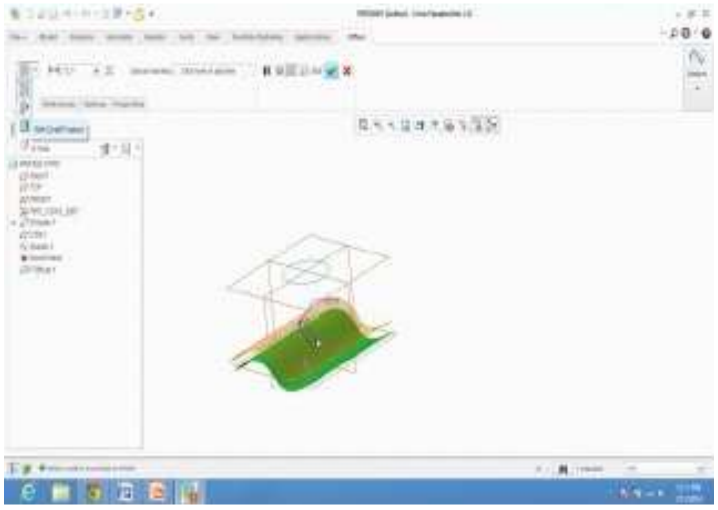


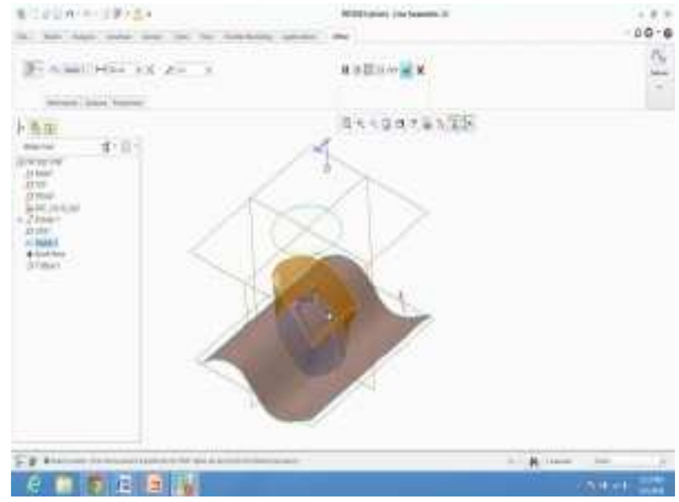


⦿ ***OFFSET***

As the name implies, this option will help you create offset surfaces or either flat or curved surfaces. We will see how to create offset surface of the **"surface-1"** of the **figure-1**. Click the Surface-1 and go to ***Offset*** and the **offset** dashboard will appear, where you have to define the offset distance, type and direction

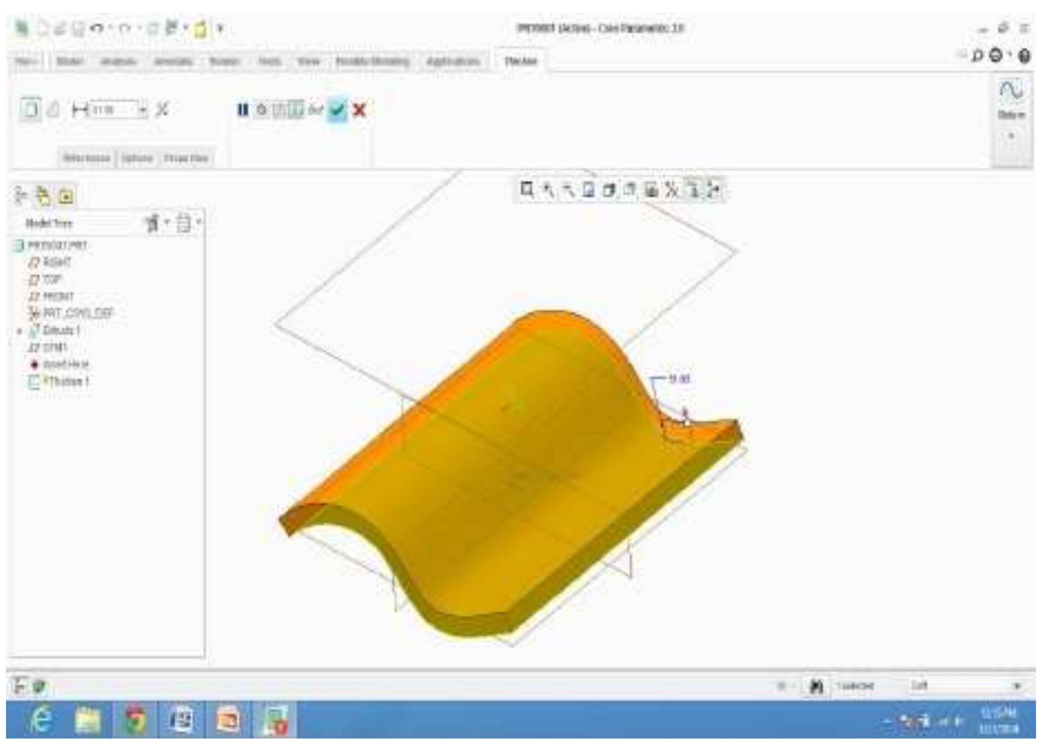






⦿ ***THICKEN***

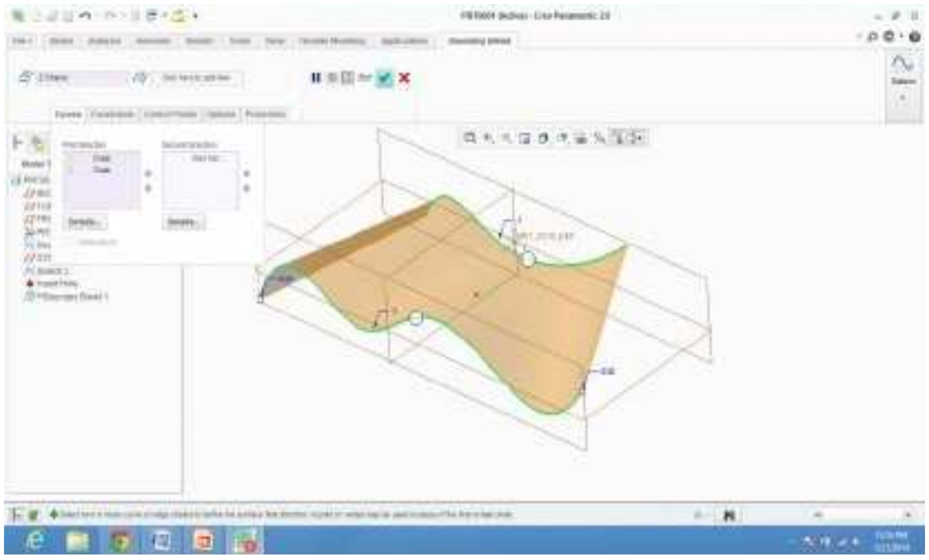
After capturing all the intricacies in the Pro-E Surface modeling you have to make it solid. For this purpose you have to use the "**Thicken"** editing option. For the above example (Figure-1) we will thicken the "**Surface-1"** and for that you have to select the Surface-1 and go to ***Edit → thicken***, the **thicken** dashboard will appear, where you have to define the thickness and direction.



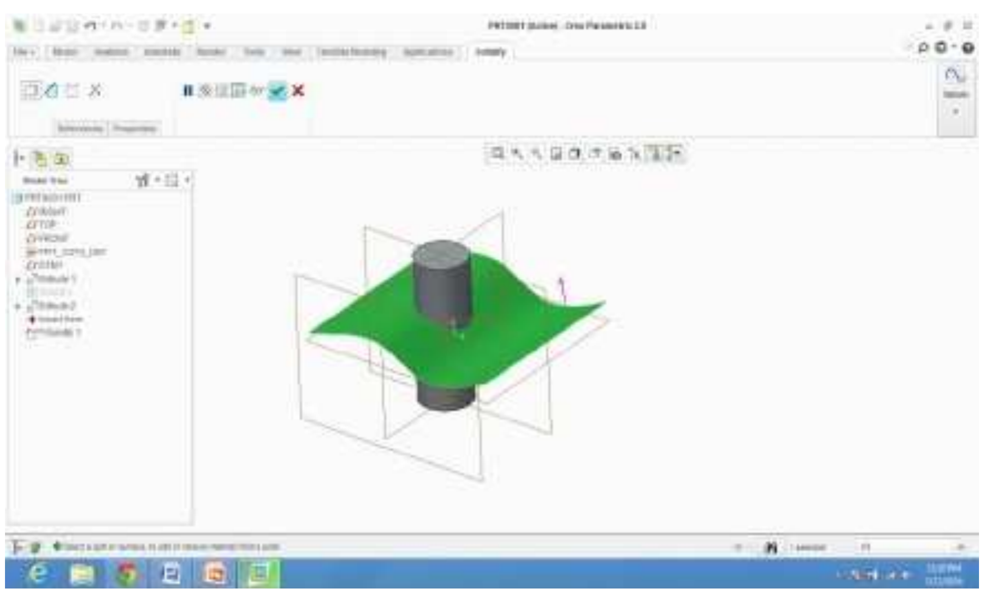
<u>BOUNDARY BLEND</u>

- The boundary blend is a Pro Engineer surface modeling command used mainly for creating smooth but irregular surfaces, where you know the surrounding 2D or 3D boundary curves of it.

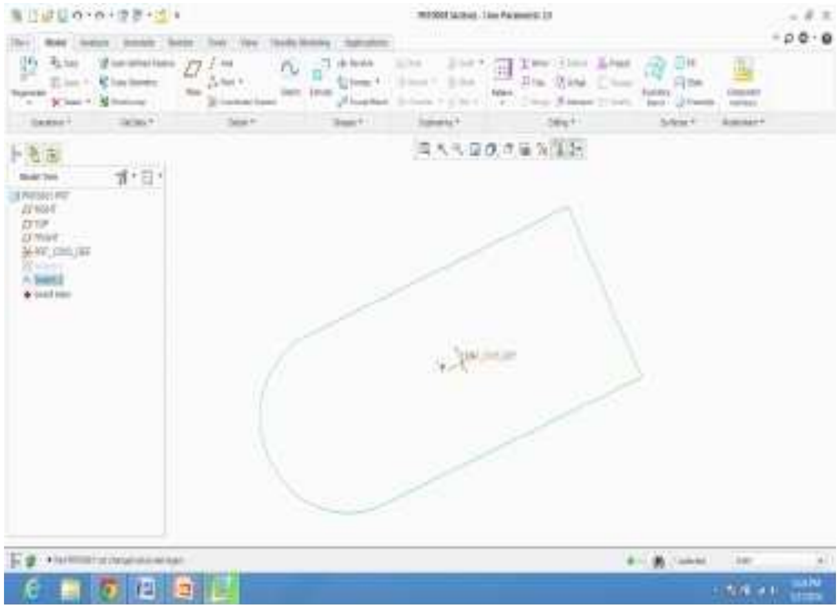
  Boundary blend is one of the **advanced surface** features (parametric surface) in CREO. Create 2 curves . Go to boundary blend > select 2 curves using ctrl .

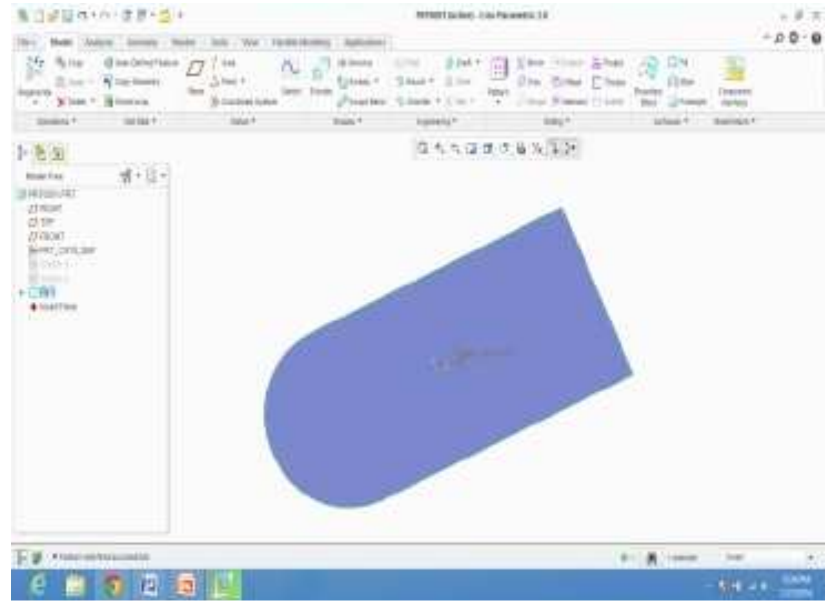


- Solidify- This tool is used to trim solids from surfaces .Create a solid and surface intersecting the solid .Select surface > goto solidify > goto remove material option from dashboard > select portion of solid to remove > click ok

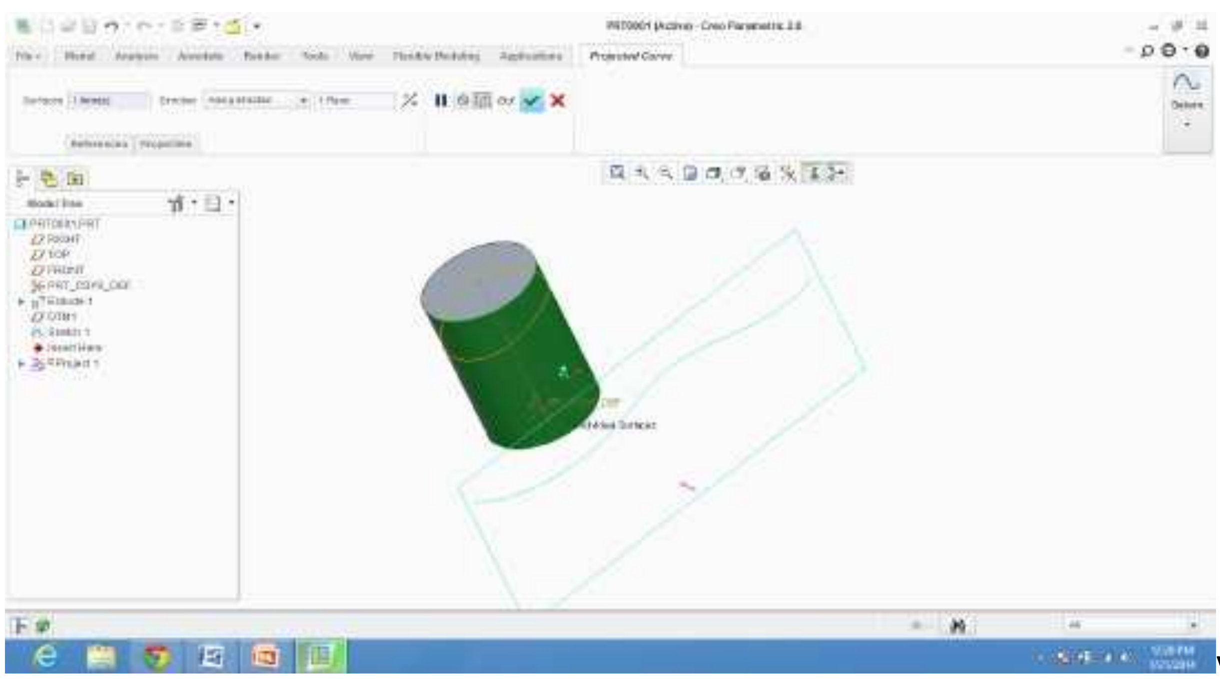


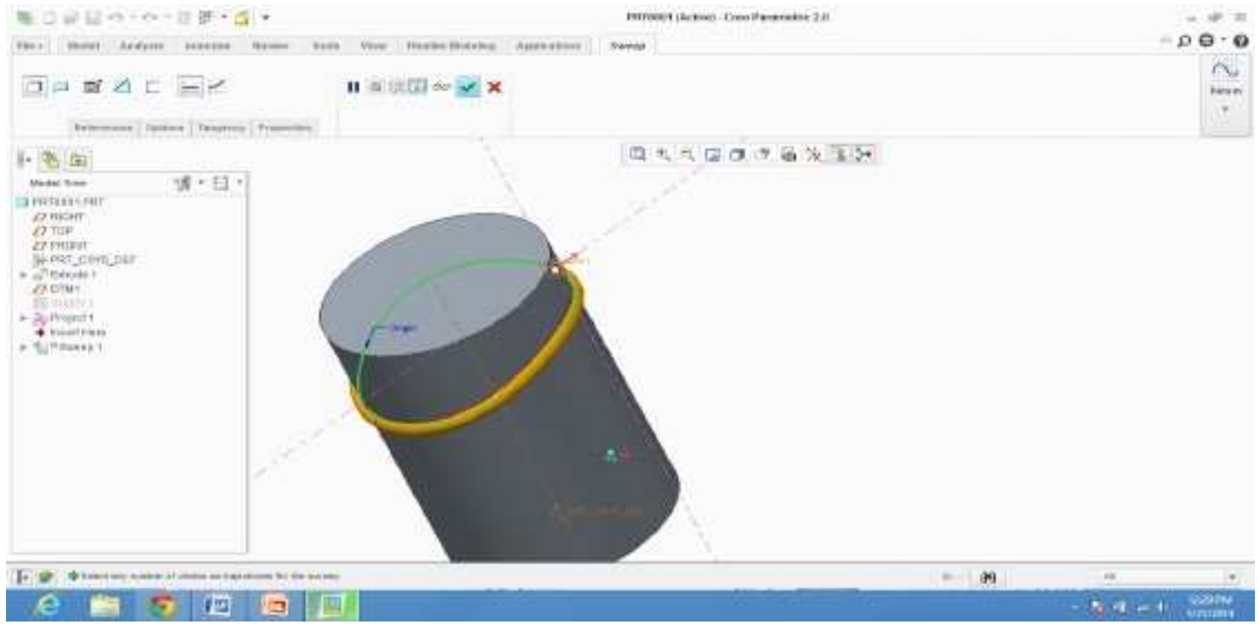
- Fill- Convert sketch to surfaces





- Project – Used to project a curve on a surface which can be later used for sweeping i.e. adding or removal of material .


v



CHAPTER 15
ASSEMBLING WITH CONSTRAINTS

An assembly is a collection of parts and other sub-assemblies that you bring together using constraints .

File Extension is . asm

There are two Approaches for assembly design.

- **TOP_DOWN APPROACH**

In the top-down assembly design approach components are created inside the assembly module therefore no need to create separate part files for the component .

This approach is **completely different** from bottom-up approach where separate components are first created then assembled by applying constraints .

NOTE : Even though components are created inside the assembly module they are saved as individual part files that can be opened and used later .

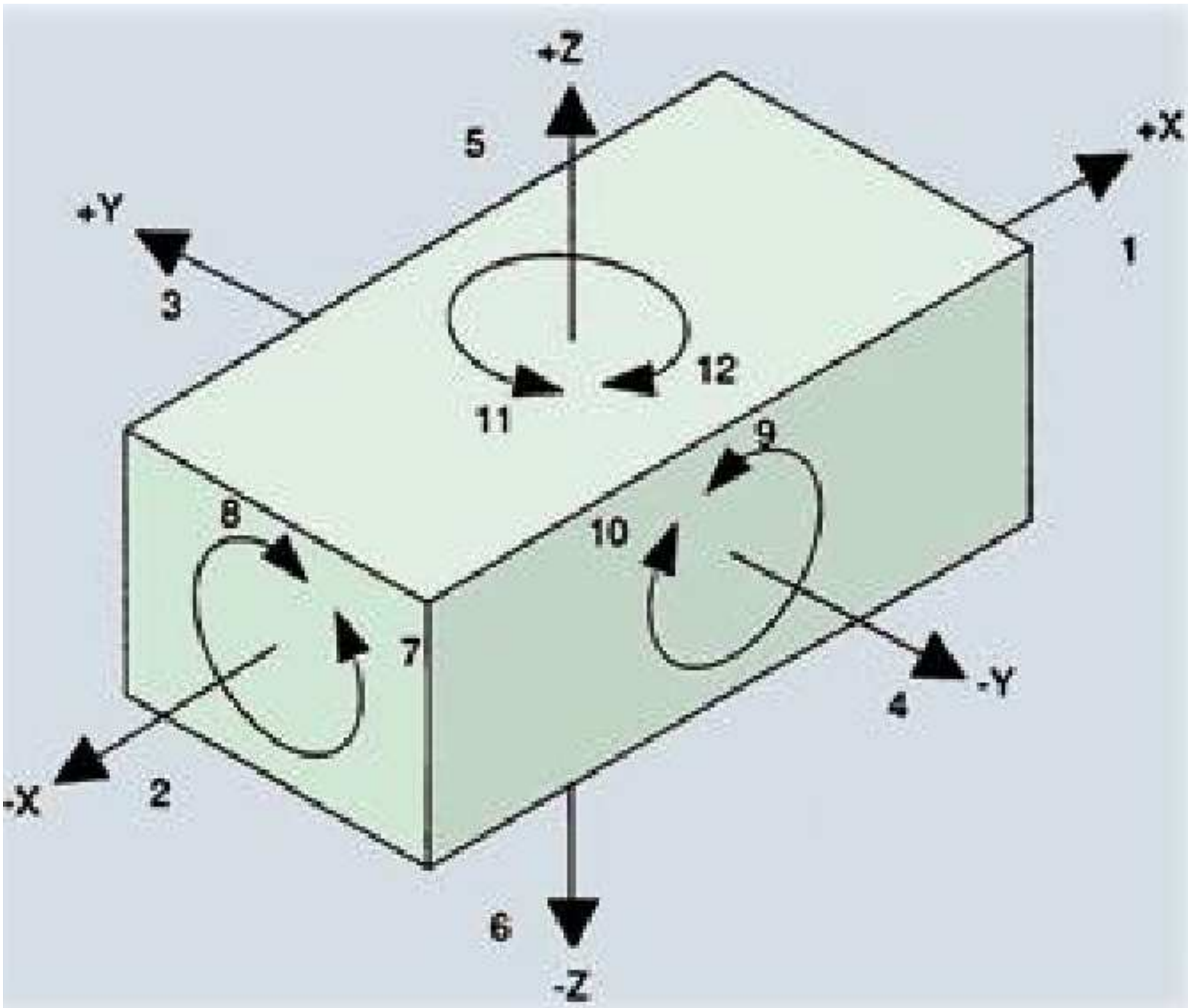
- **BOTTOM UP APPROACH**

A **bottom-up** approach is most common as it is the traditional and most logical approach. In this approach we create the individual parts independently, insert them into an assembly and use the mating condition to locate and orient them in the assembly as required by the assembly design.

How design intent is captured in assemblies ?

Ans: By restricting degrees of freedom.

There are 6 degrees of freedom in all:



Degrees of freedom are restricted by applying assembly constraints.

- Select component reference
- Select assembly reference

Types of constraints in assemblies :-

AUTOMATIC

DISTANCE

ANGLE OFFSET

PARALLEL

COINCIDENT

NORMAL

COPLANAR

CENTERED

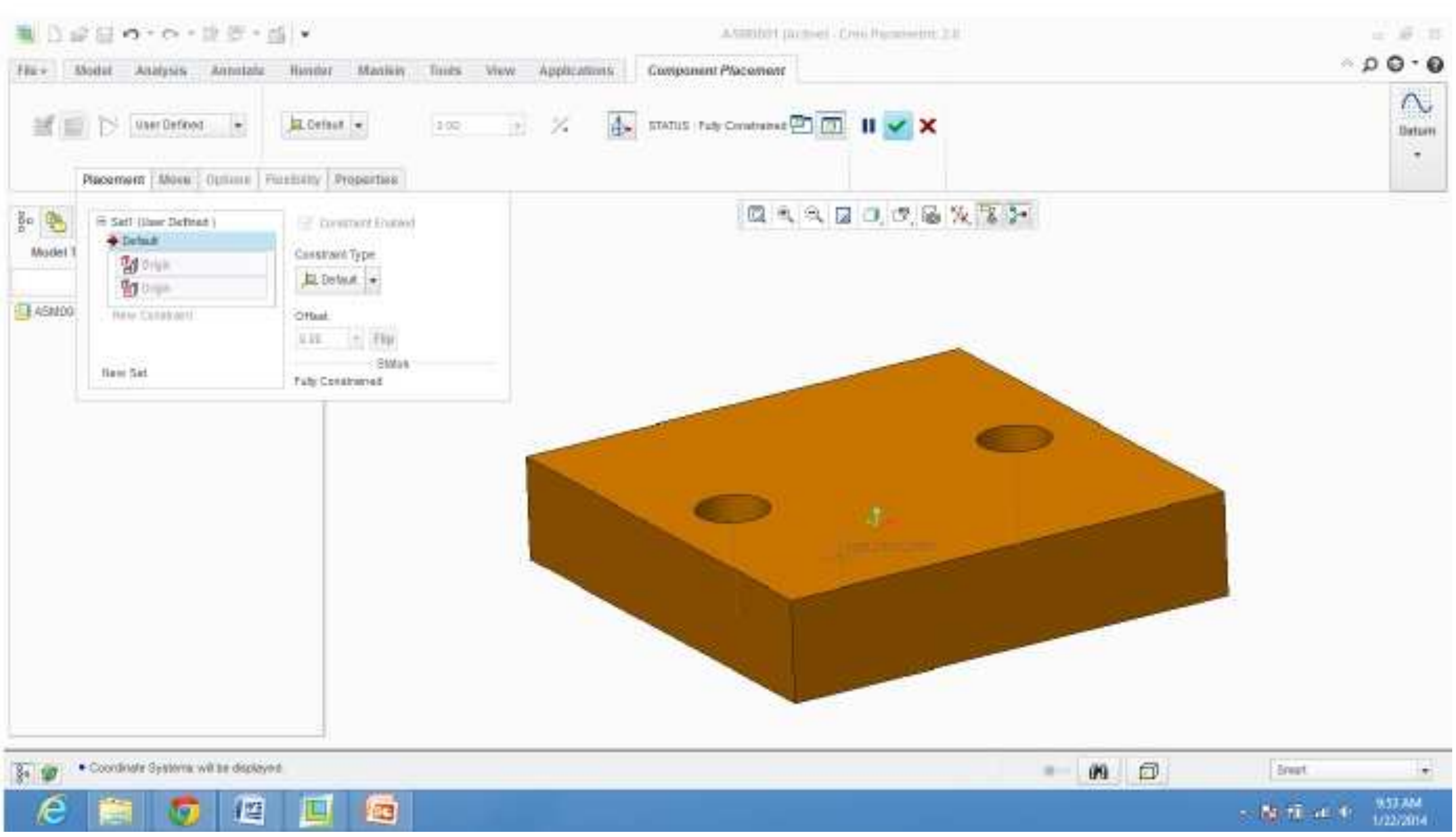
TANGENT

FIX

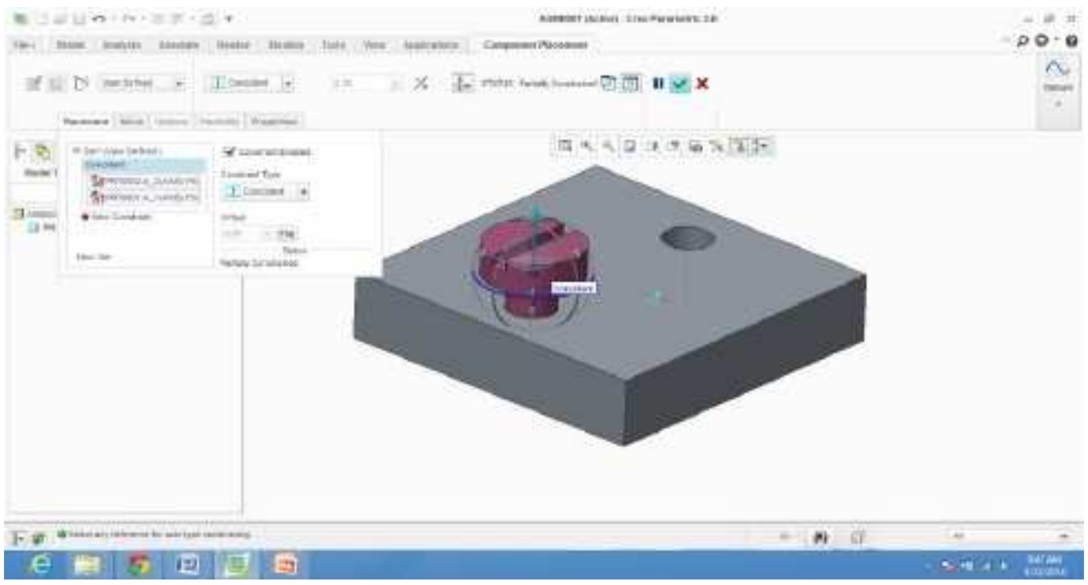
DEFAULT

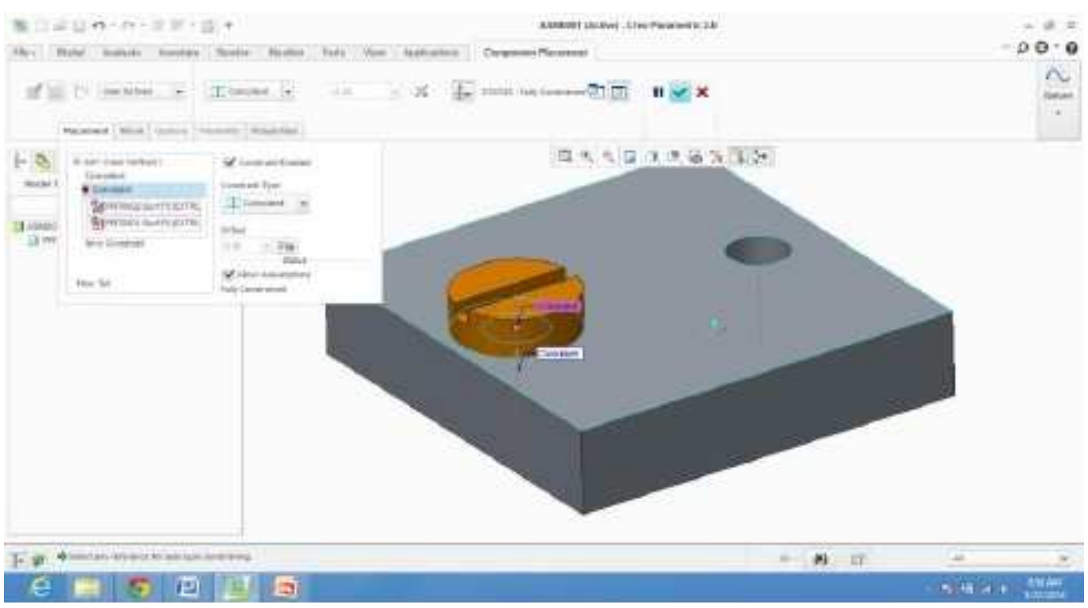
For most of cases we will use default , coincident and distance constraints.

<u>Default constraint:</u> Coordinate of part merges with coordinate of assembly . It as a standard practice to assemble first component of assembly using default constraints .

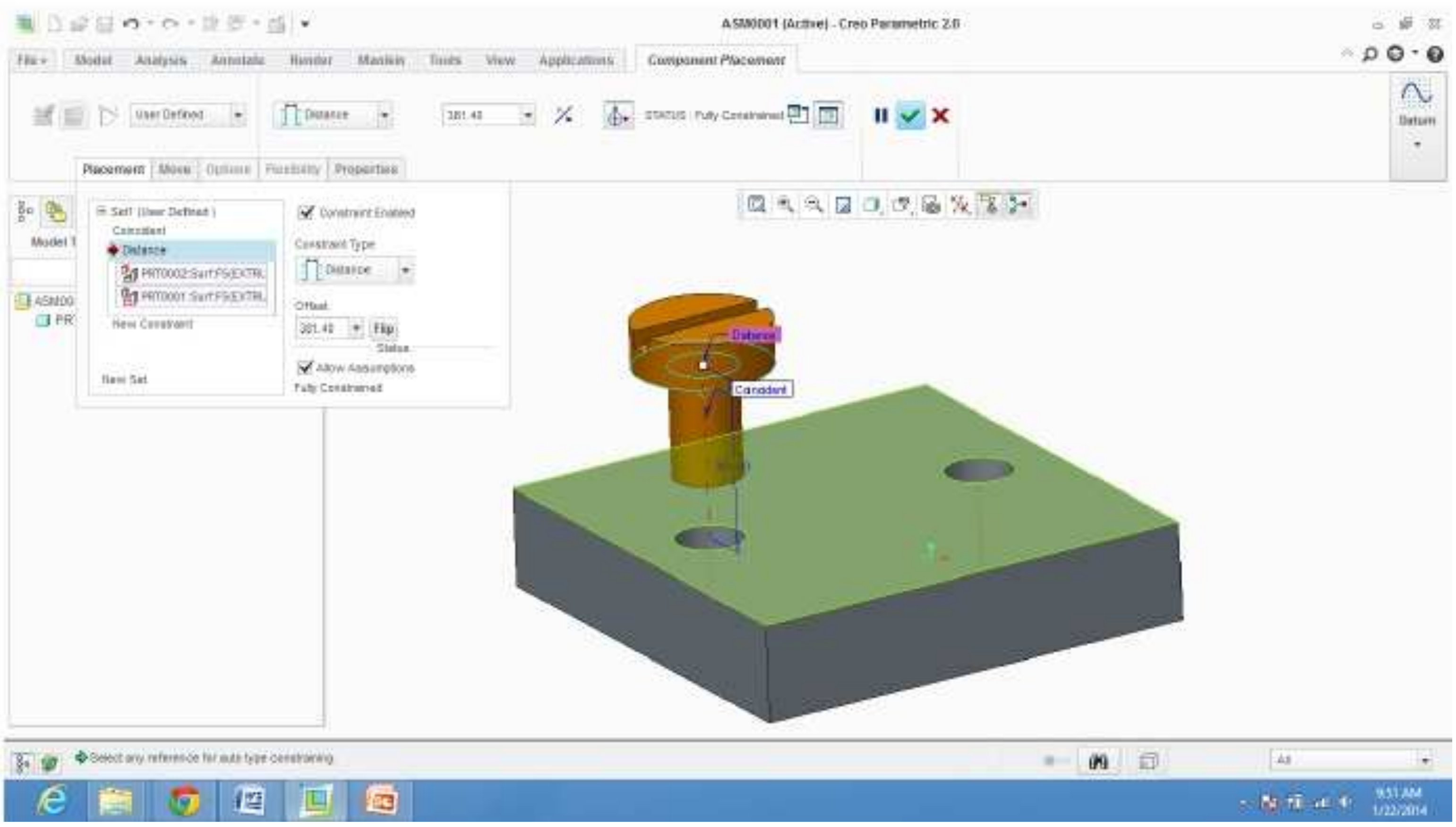


<u>Coincident constraint</u>: This constraint enables you to position two planar surfaces or planes to lie on a same plane (coplanar) , or make 2 axis coaxial .





Distance constraint: This constraint enables you to place 2 planar surface at a distance to each other .



CHAPTER 15

EXPLODING , CREATING CROSS SECTIONS ,FAMILY TABLE , STYLE , CUSTOMIZED ORIENTAIONS

Family table

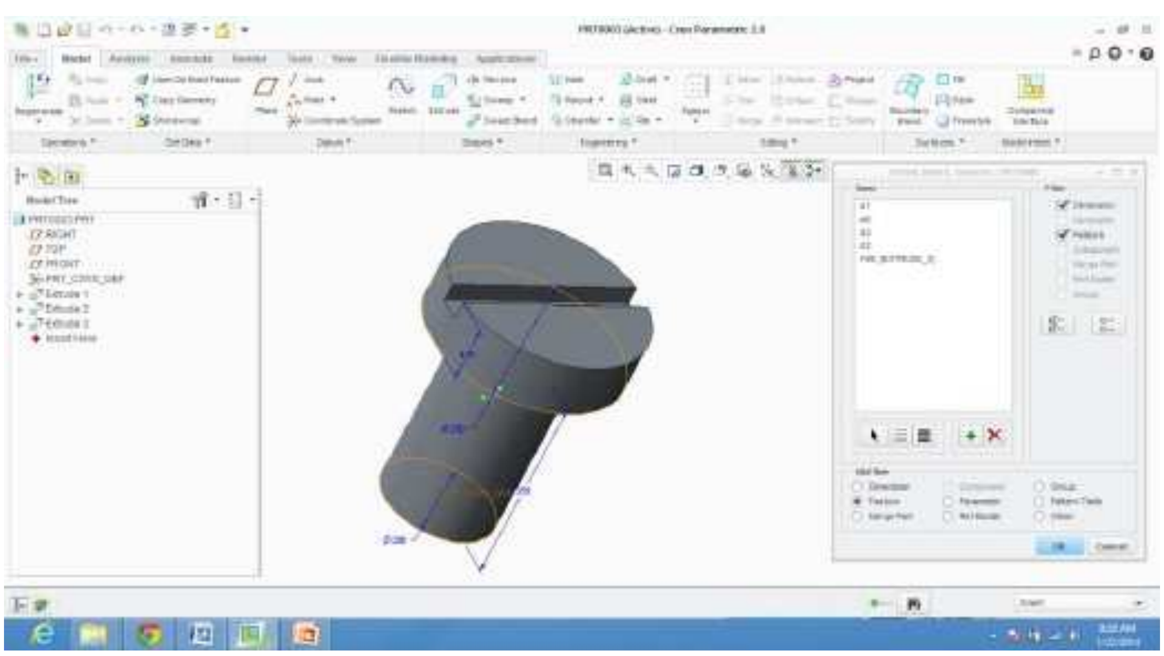
Family tables are a collection of parts , assemblies , or features that are similar , but deviate slightly in some aspect , such as size . For example, bolts of a certain type , though different in size , all look alike and perform same function . Parts in family tables are also known as table-driven parts.

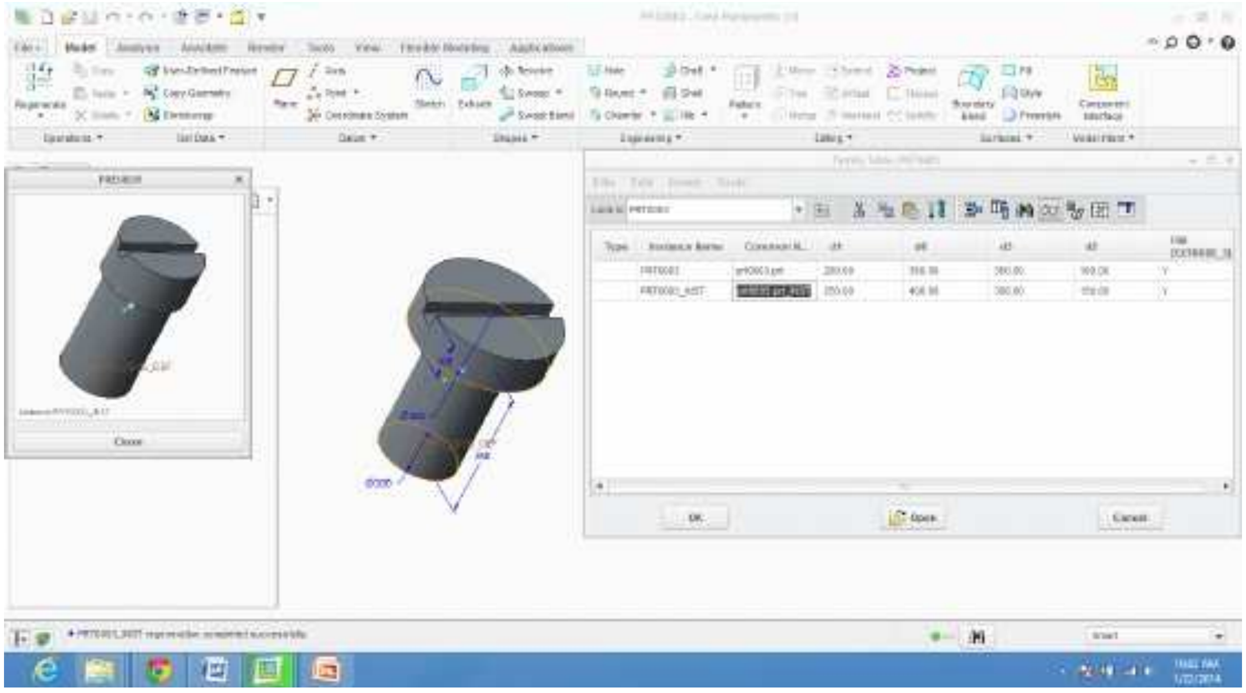
Rows : Family table rows contain the generic or initial model and instances of parts and their corresponding values . The generic model is the first row in the family table .

Columns: The family table columns are used to specify the items in the generic model that can be varied in the instance .

Family table uses

1. Create and store large number of objects simply and compactly within a single model.
2. Save time and effort by standardizing part generation
3. Generate variations of a part from one part file instead of having to model each one

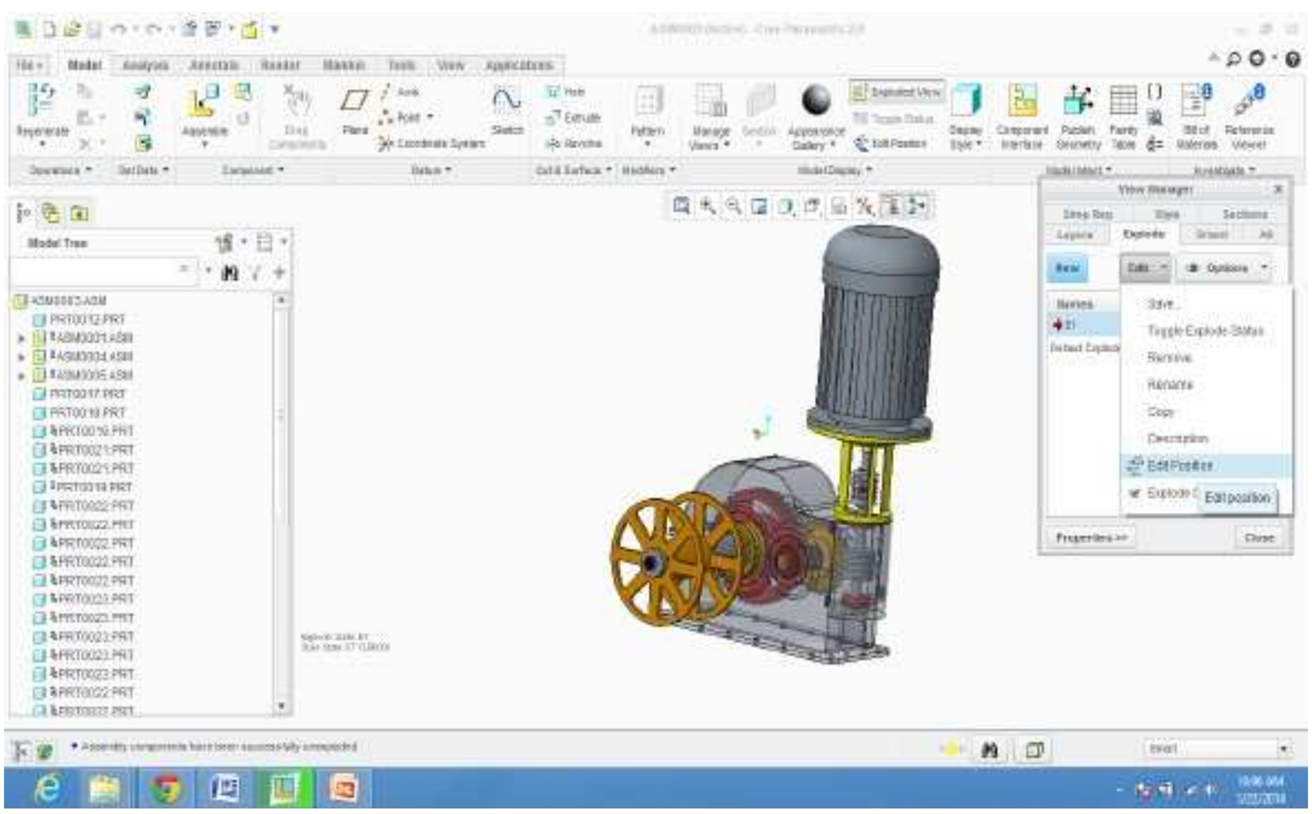


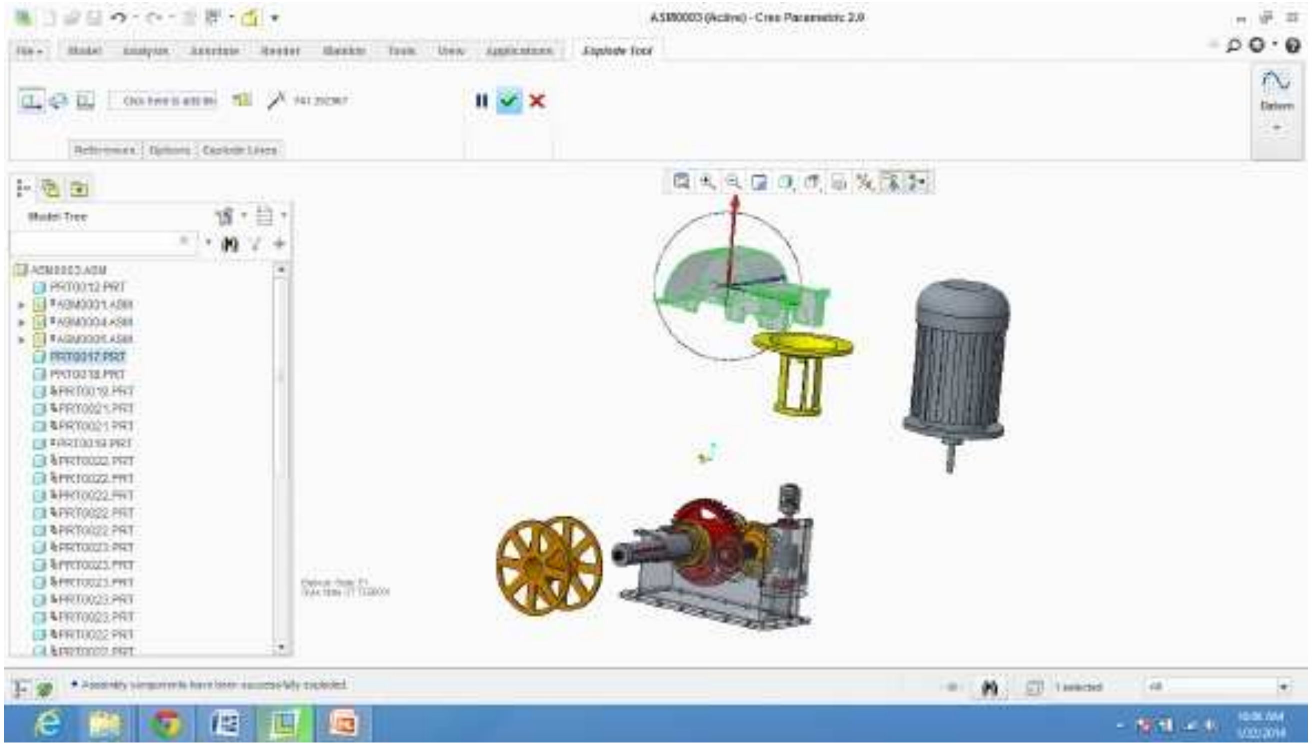


CREATING CUSTOMIZED EXPLODED STATE

Software provides default exploded state by itself , in order to get customized exploded sate of assembly software provides VIEW MANAGER tool in graphics toolbar.

Goto view manager > explode > name new exploded state >edit > edit position > manage position of components > click ok > + sign appears beside name > rt click > save

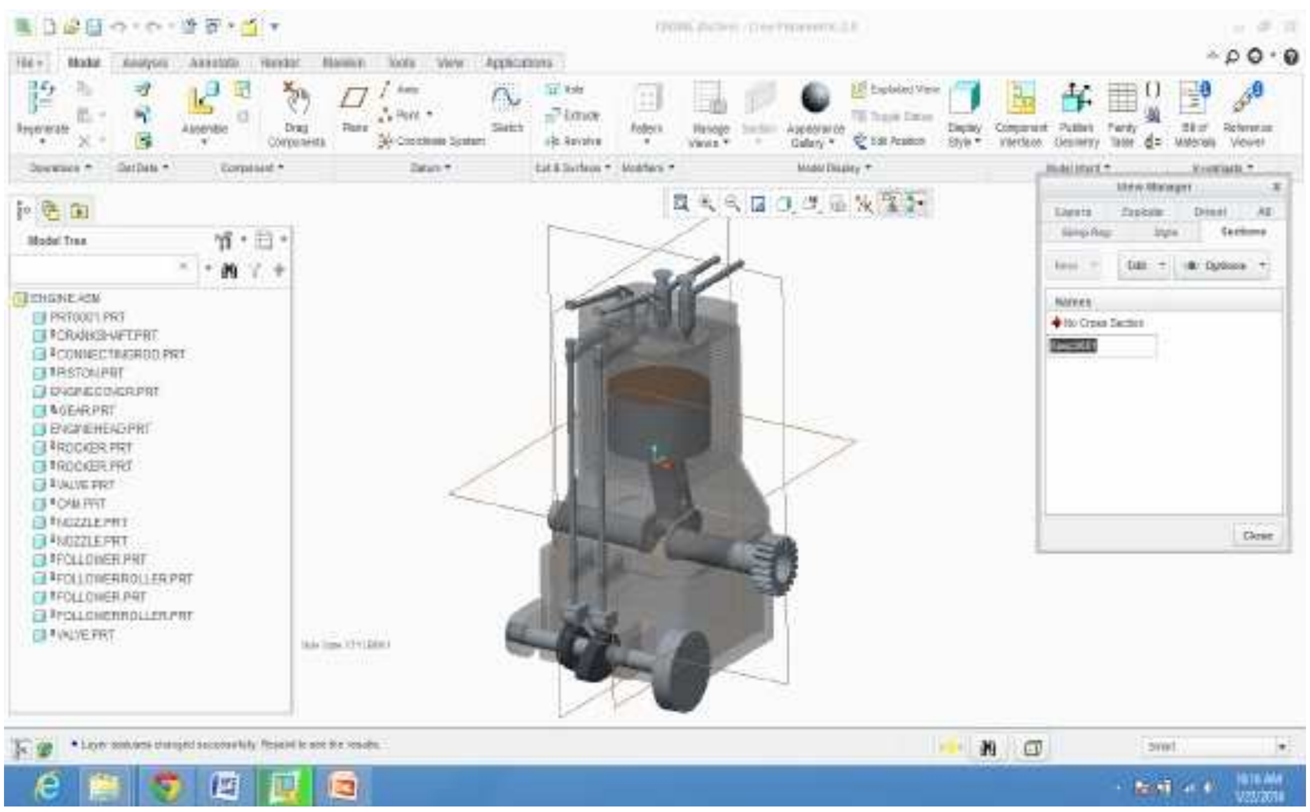


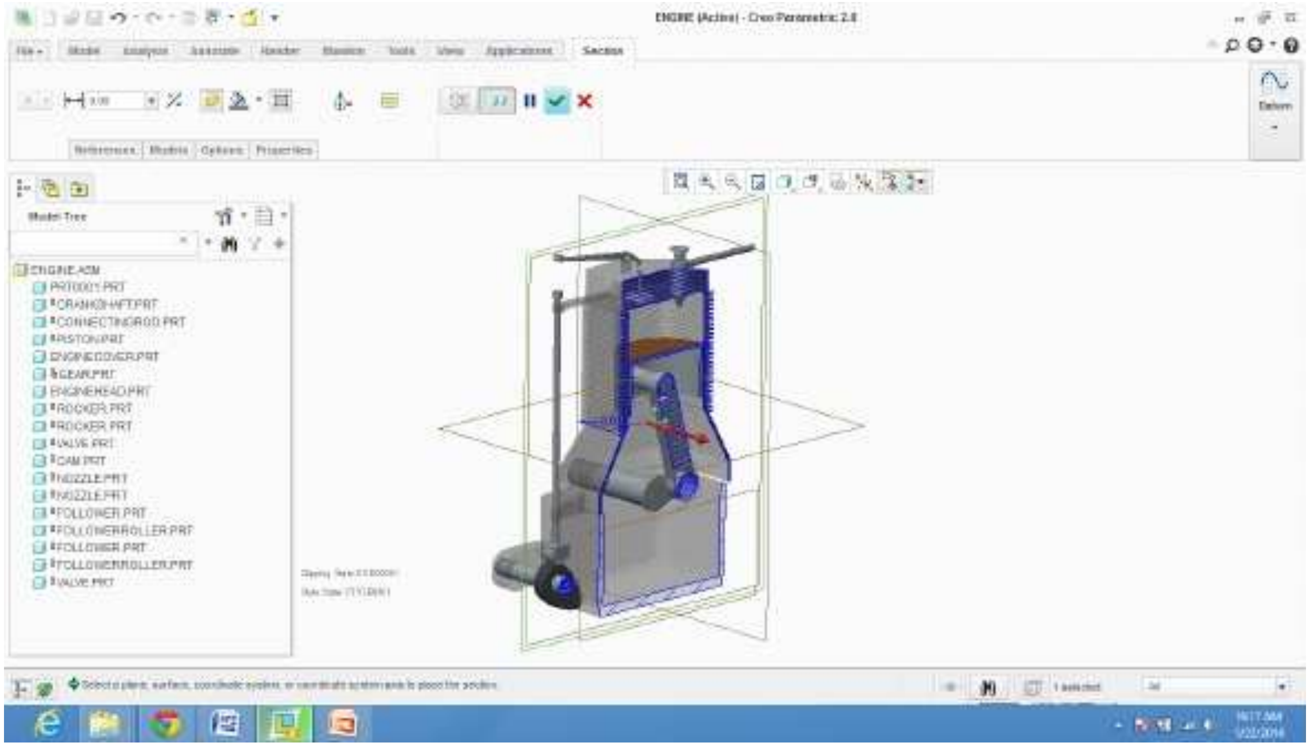


CREATING ASSEMBLY CROSS SECTIONS

Planar-The cross section is defined by a datum plane 0r planar surface intersecting the model.

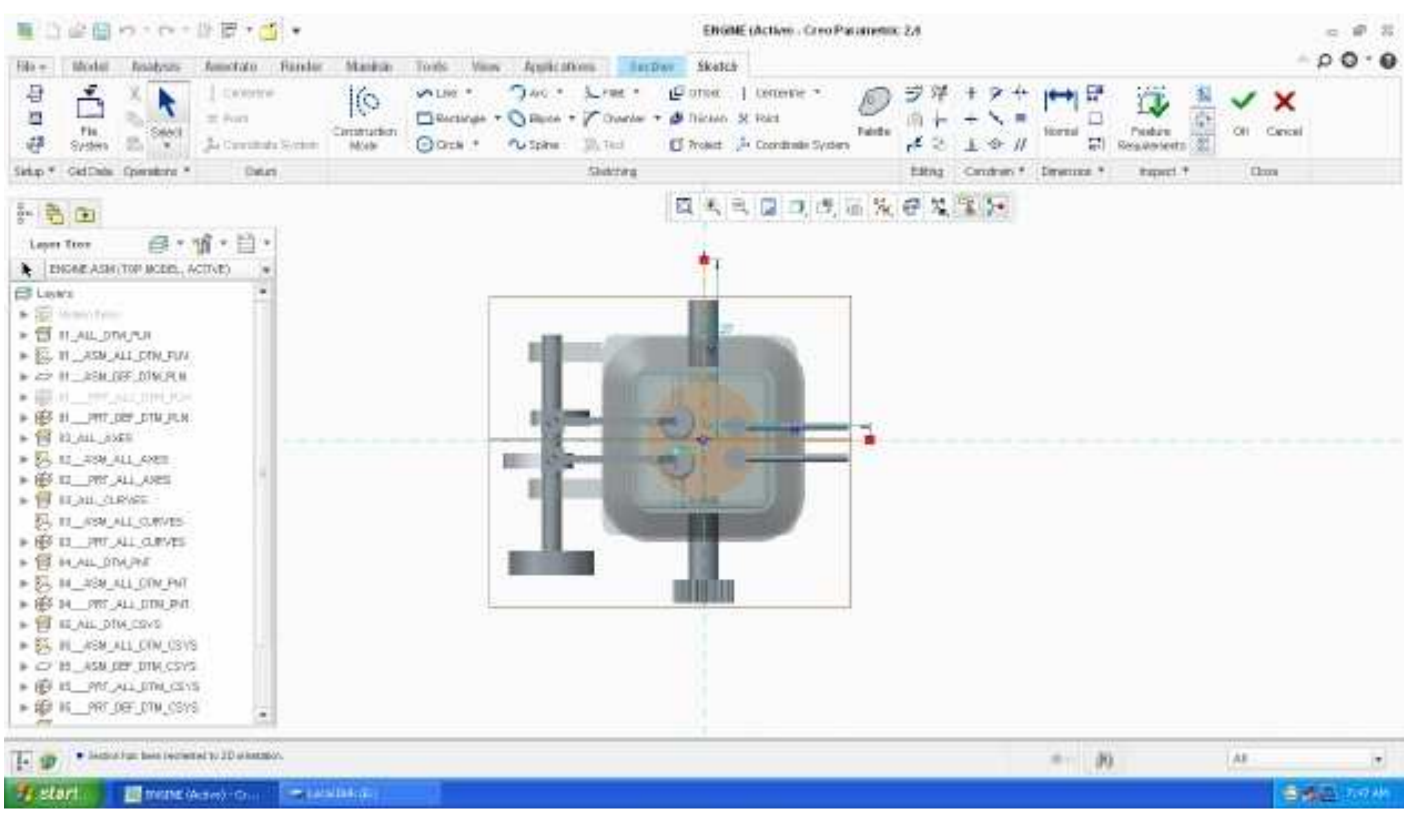
Goto view manager> sections > new> planar >name the section > select plane (you can turn ON the hatching)

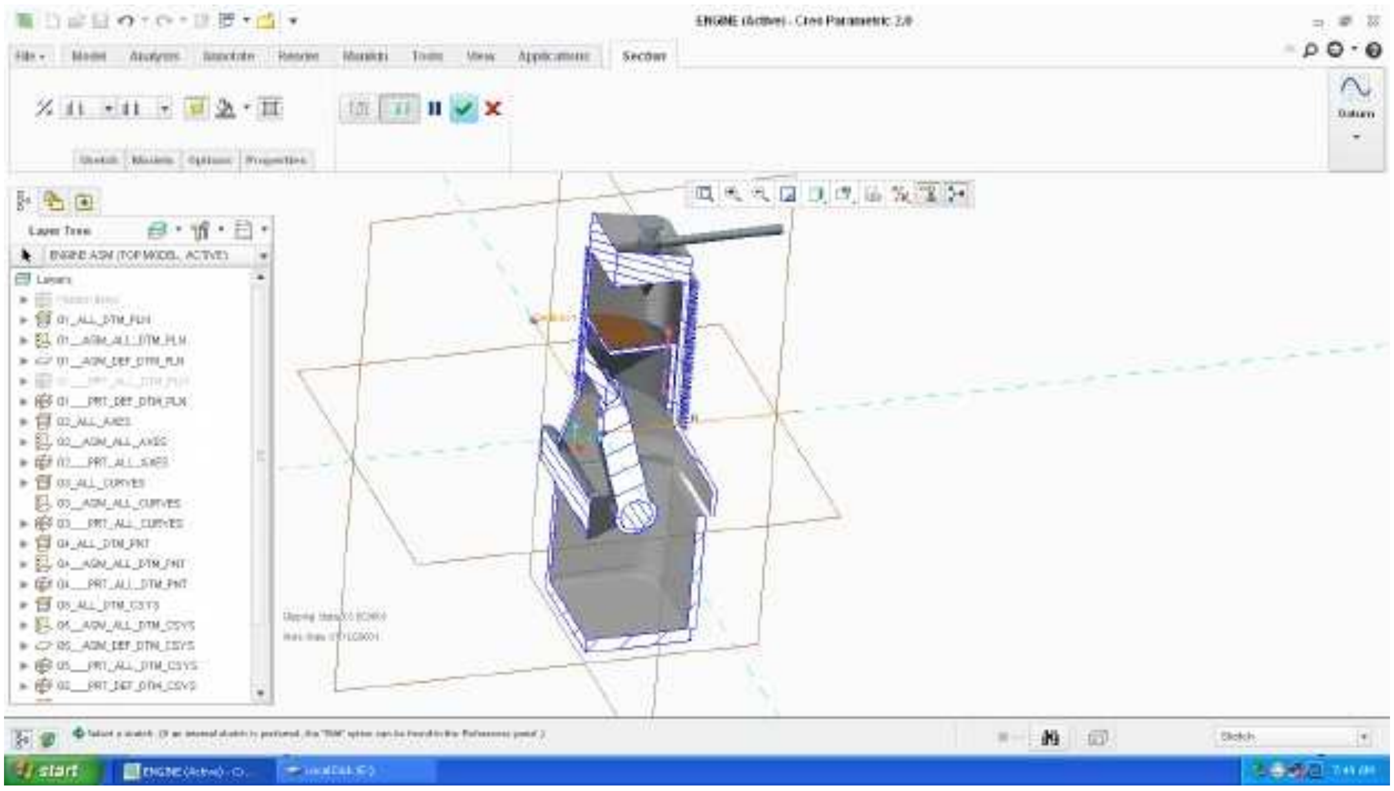




Offset- An offset cross section uses a sketched cut line to intersect the assembly .The cut line shape is then extruded in one or both directions from the sketch plane.

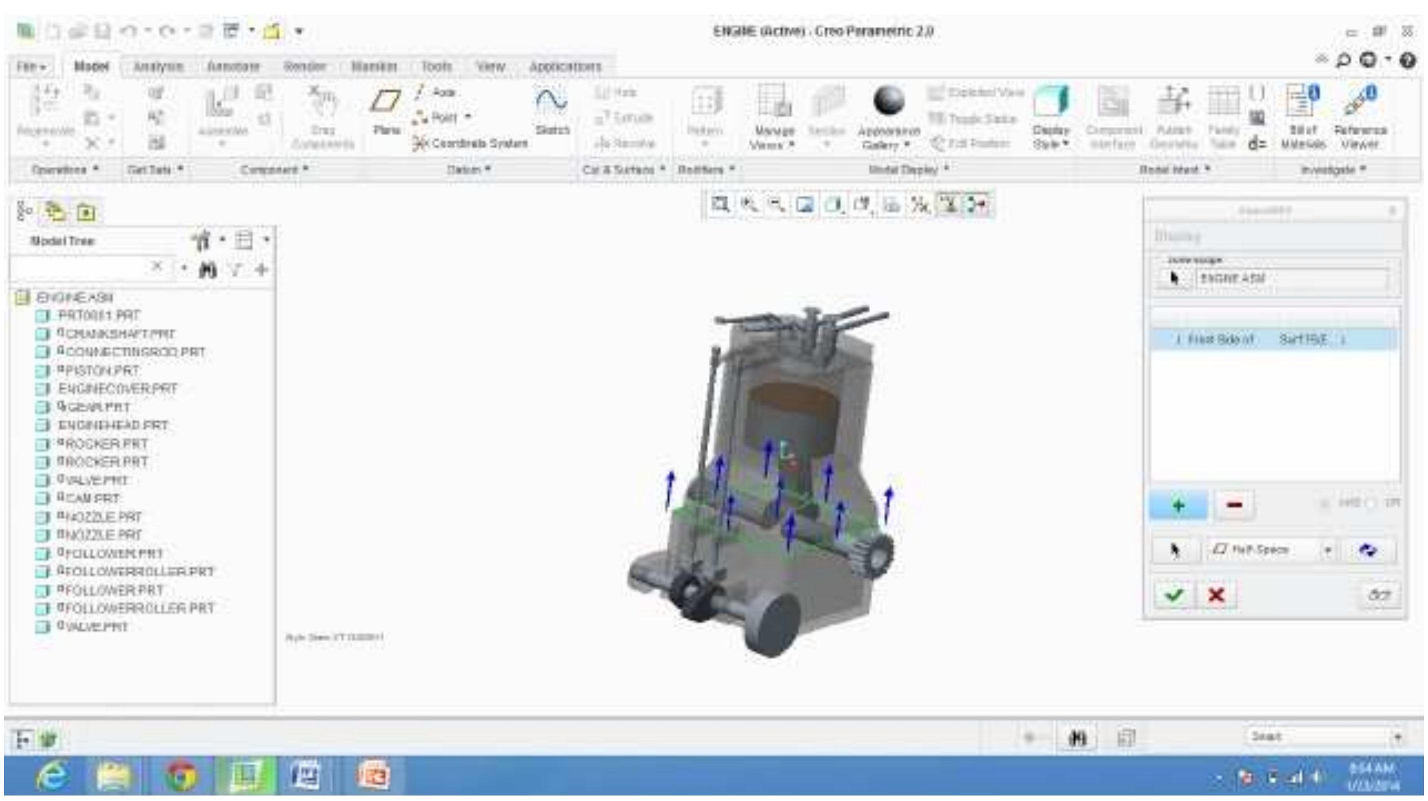
Goto view manager > sections > new > offset >name the section > create sketch > done





Zone-Define a region within an assembly that can be used to select a number of intersecting components.

Goto view manager > sections > new > zone > name the section > select plane > click on done



CREATING CUSTOMIZED DISPLAY STYLES

Display style is used to control the display of models in CREO session.

Types of display styles

Wireframe

Hidden Line

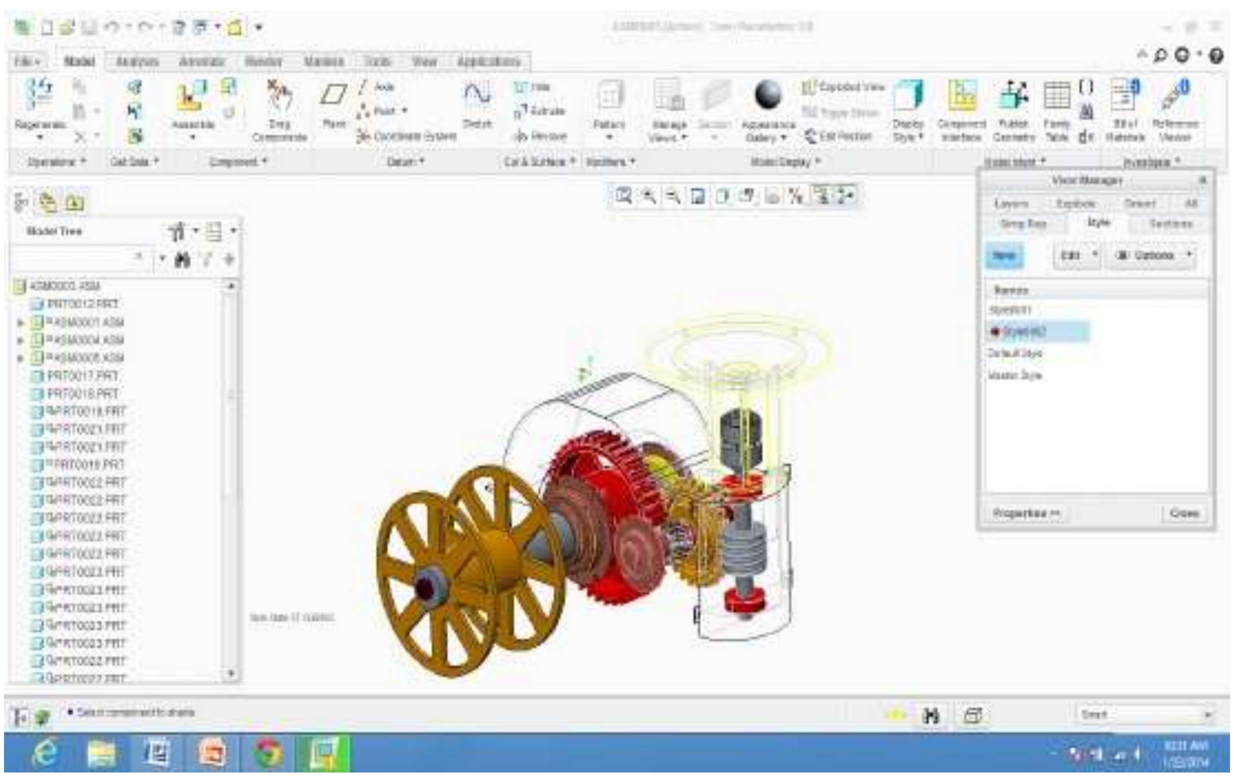
No Hidden

Shaded

Transparent

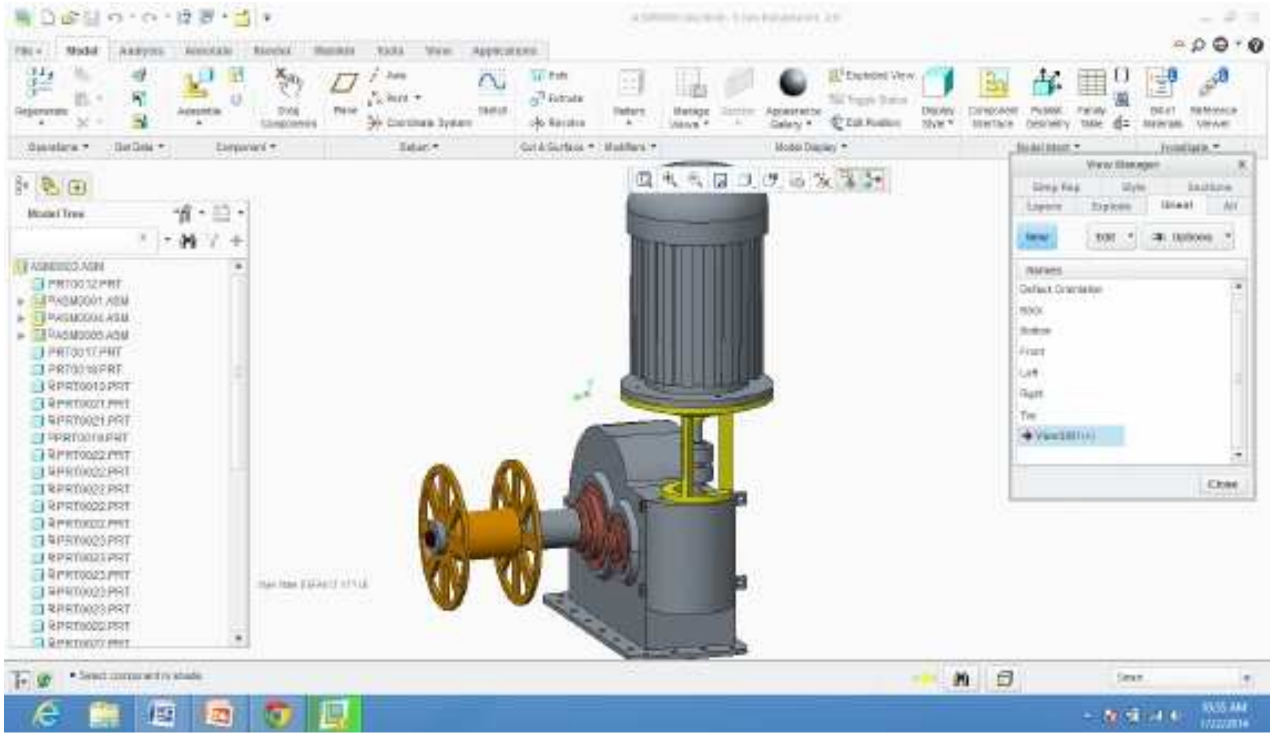
Blank

Goto view manager > style> new > name the style> select components to be blanked , wireframe , hidden , no hidden , shaded , transparent > click ok



CUSTOMIZED ORIENTATION

Goto view manager > orient > new > name the view > now orient the model in graphics window > + sign appears beside icon > rt click on icon > click save



Customized exploded state , styles , orientation are all required for drafting and animation .

CHAPTER 16
DRAFTING IN CREO

- Drafting is a Documentation of the Modeling.

  In drawing mode you can create customized documentation for the component process Views of each step can be placed, component display controlled based on their status in the step, and report tables (BOM) .

Step 1: Adding models in drafting sheet

Goto drawing model> add model > add components as well as assemblies in list > done

Now we have added models in the drafting module . Our next step is to set models to be displayed on the sheet .

Step 2: Setting models

Goto drawing model> set model > select component to be displayed > done

Step 3: Setting up general view . By default general view is the first view that is set .

Goto general view > left click on screen (model will be displayed)

Manage the following :

View type

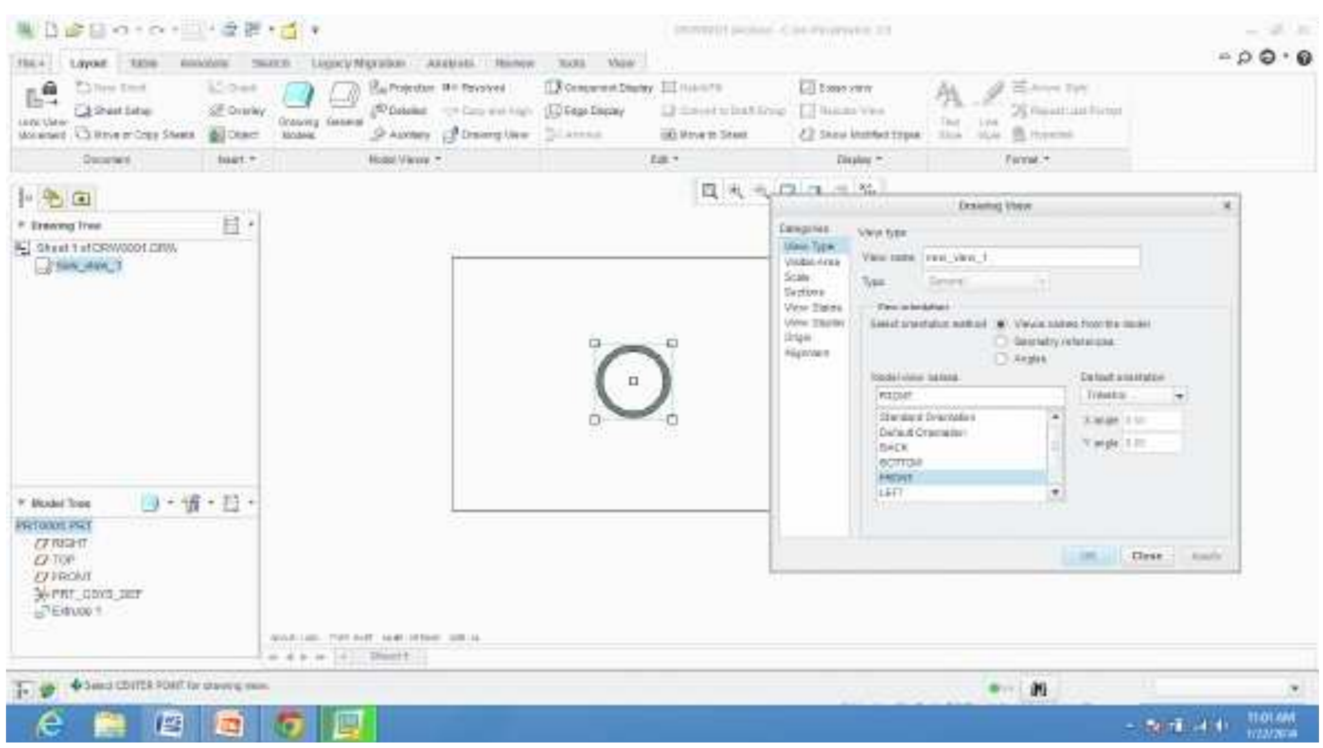
Visible area

Scale

Sections

View states

View display



How to add sectional views ?

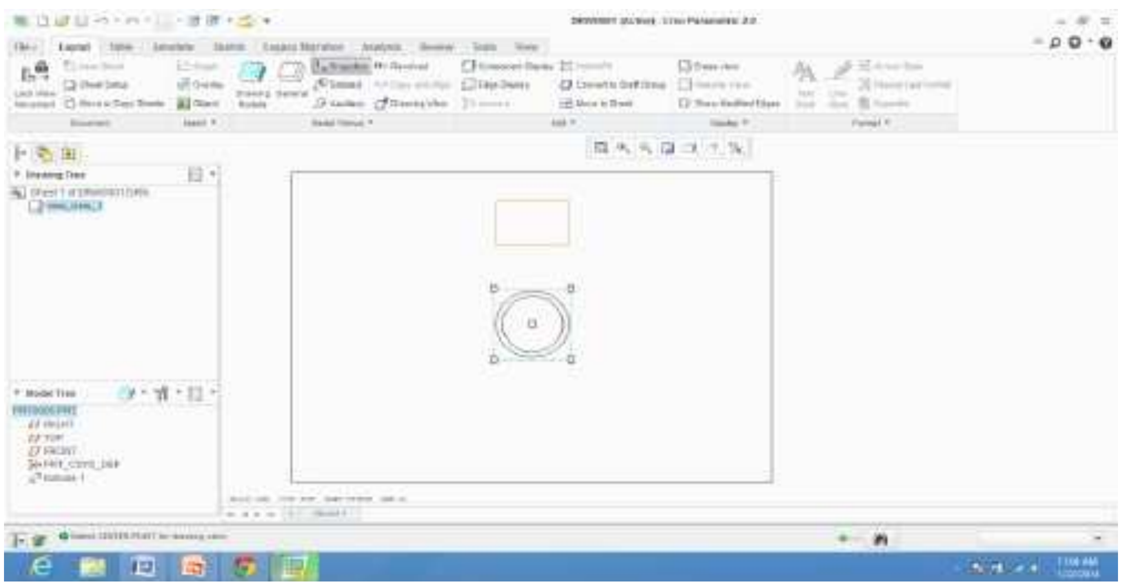
Open model> view manager > sections > new > planar > name the section > select plane > hatch > done

Goto general view> sections > 2d cross sections > add > select cross section that is ticked > apply > ok

<u>Step 4</u>: Creating projection view

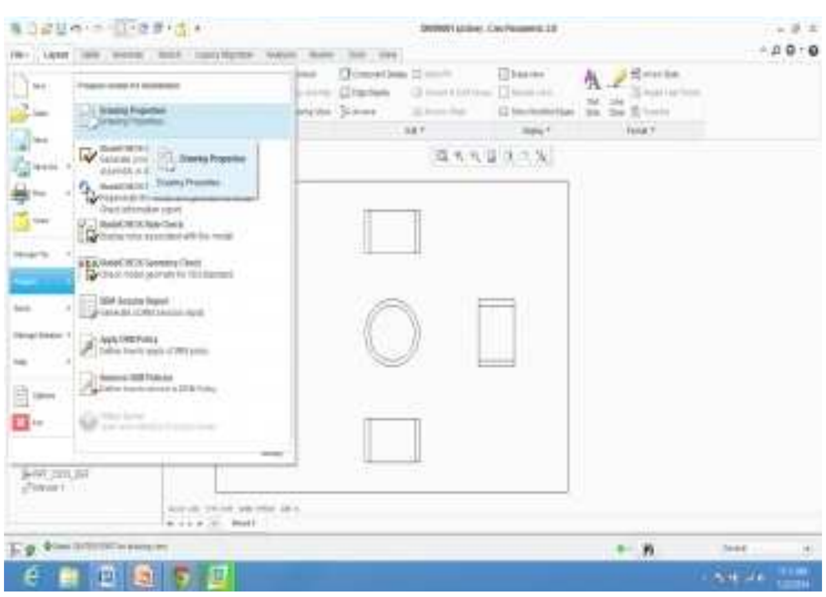
Click on model >projection > create top ,bottom, left , right projection views

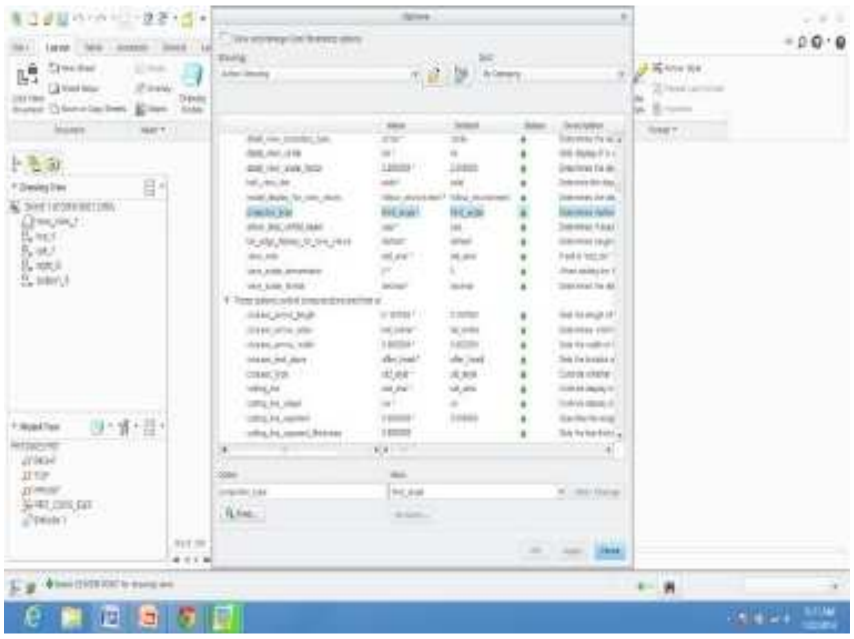
There are 2 types of projections viz. 1st angle projection(used in Europe) and 3rd angle projection(used in USA)



You can change setting from 1st angle to 3rd angle or vice versa .

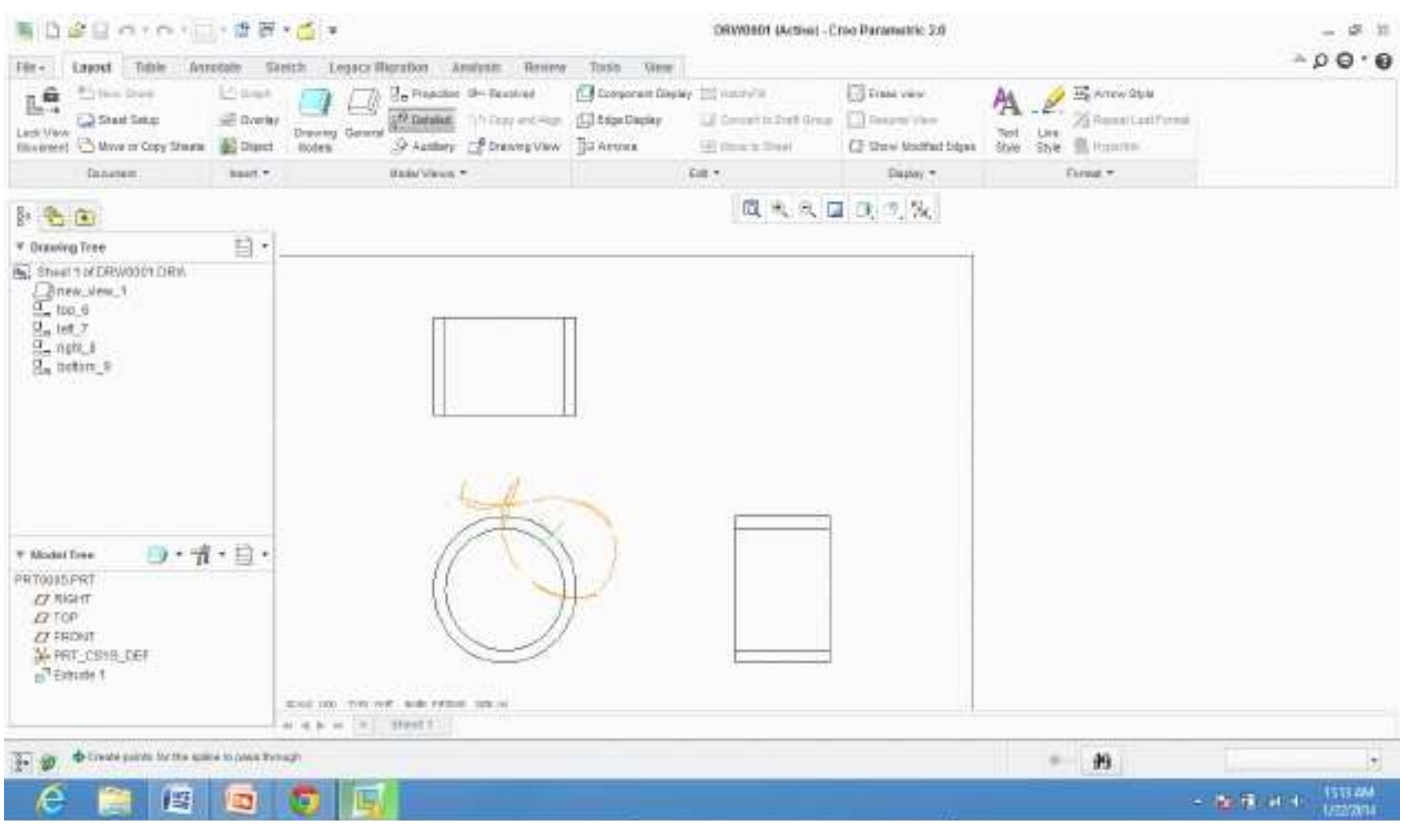
Goto file > prepare > drawing properties> detail options > change> change angle of projection from 1st to 3rd or 3rd to first > click on add/change> ok

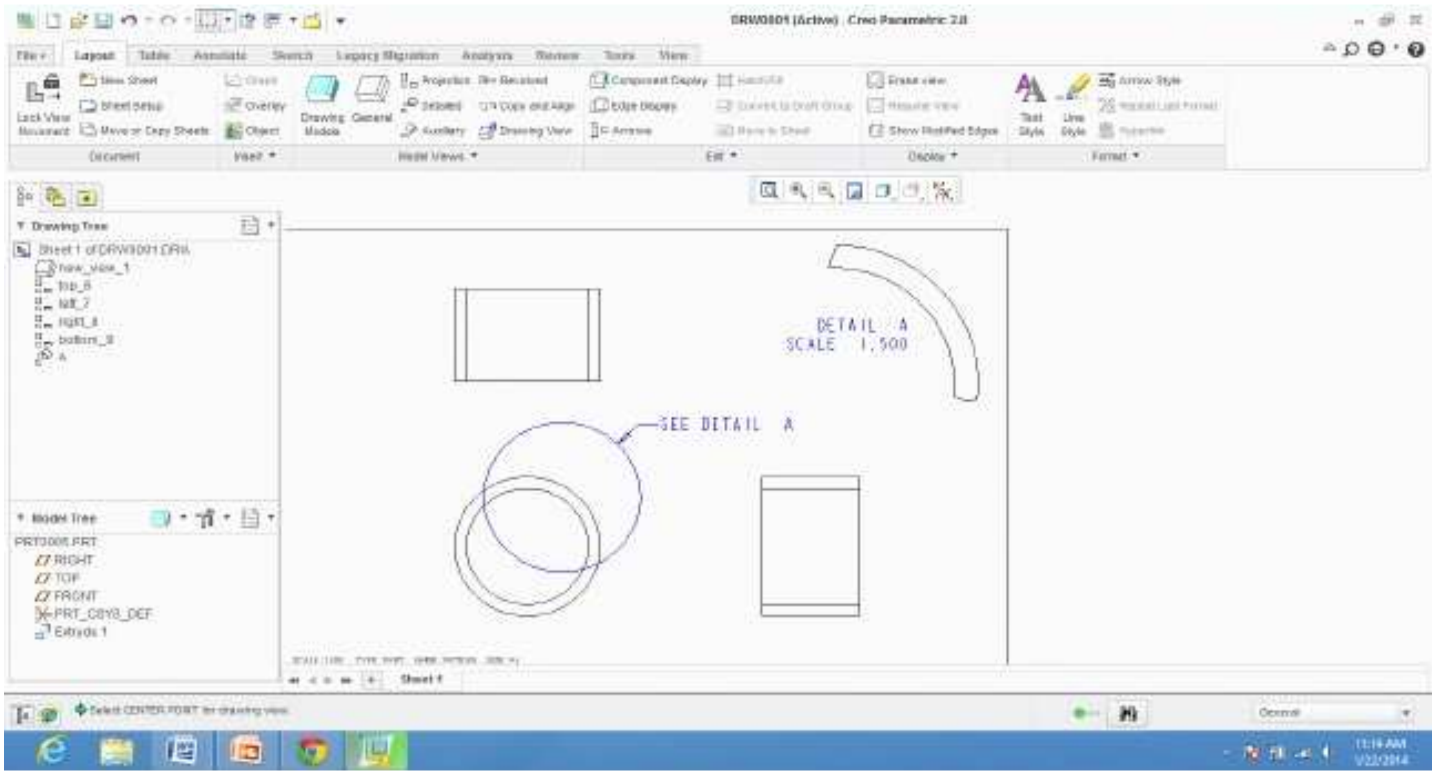




Step 5: Creating detailed view

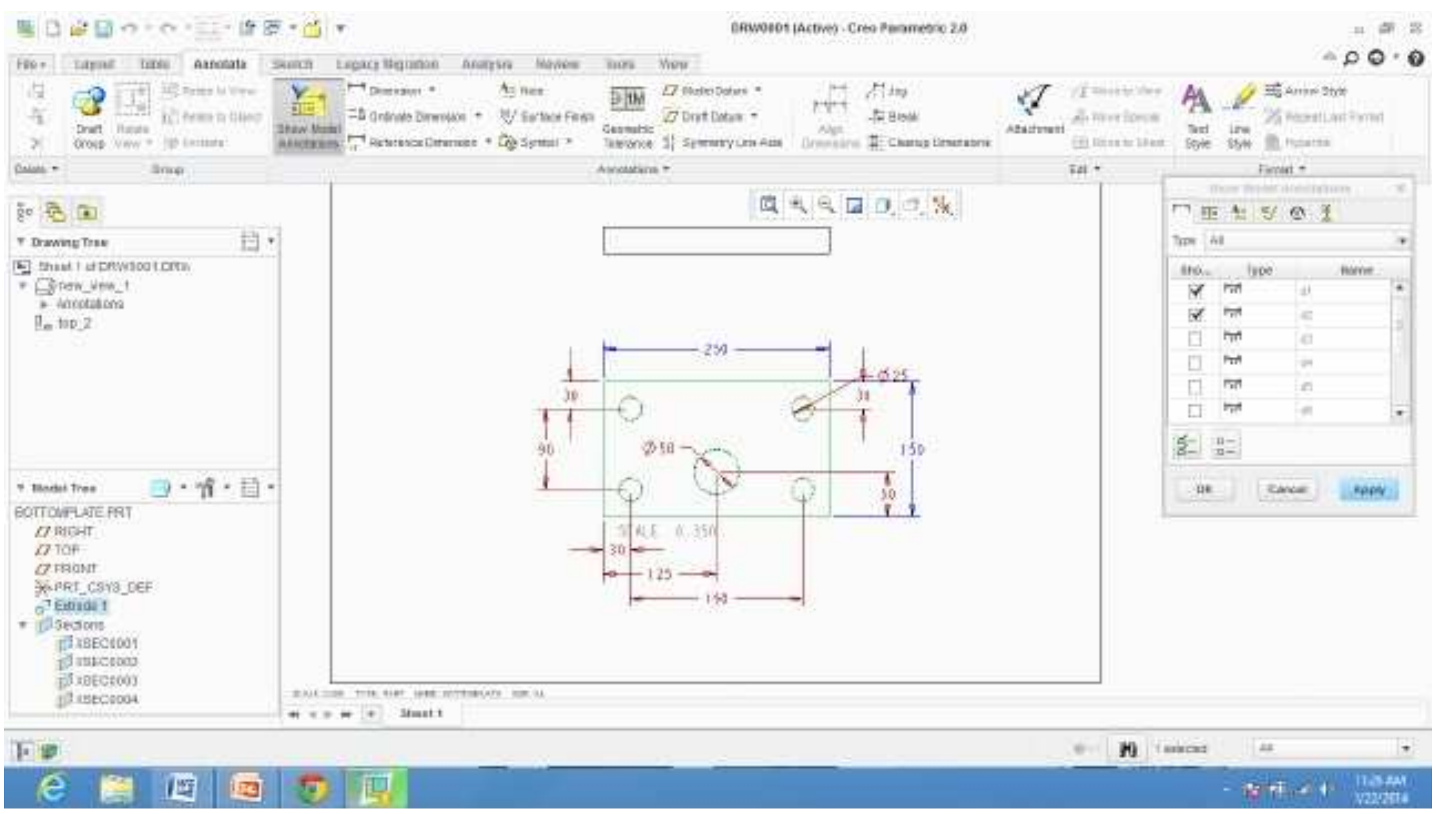
Click on detailed icon from drawing ribbon > click on entity (blip will appear > create spline around blip (see detail) will be displayed > click on any portion of graphics window to display detailed view





<u>Step 6</u>: Dimensioning

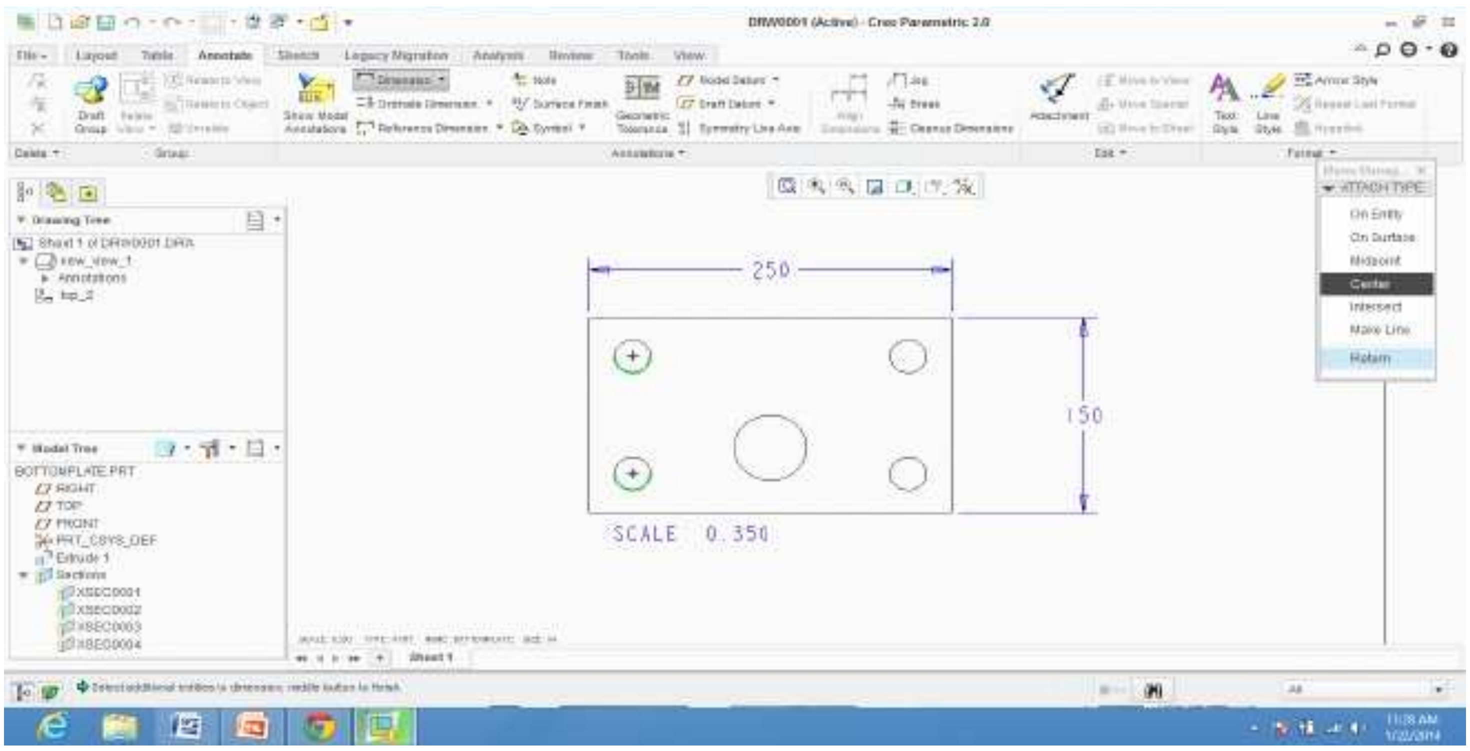
Goto annotate tab> show model annotations > click on the view> select dimensions to be displayed > apply

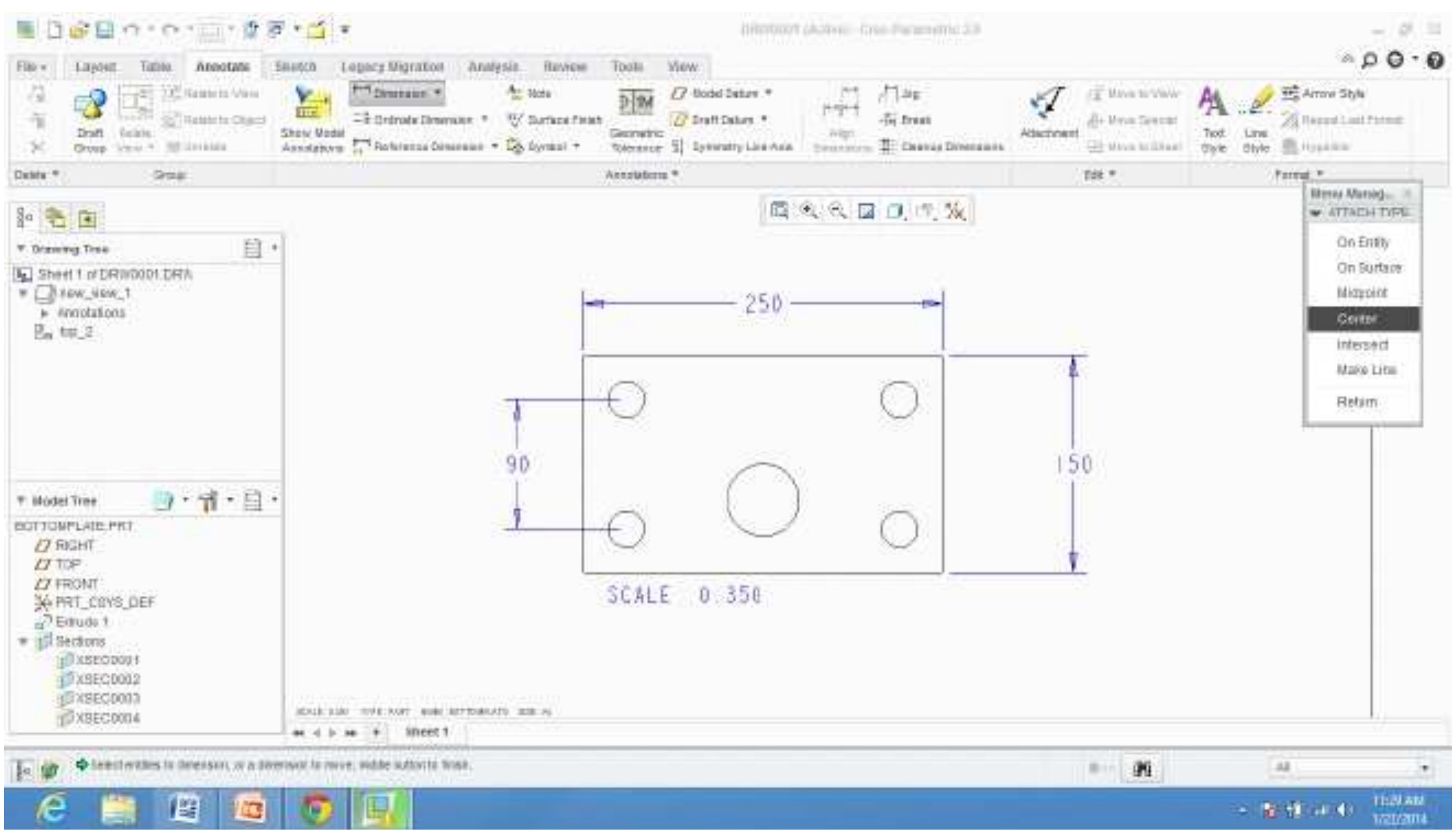


Sometimes our intended dimensions are not displayed by ‘show model annotations ‘ . In this case our step would be

Goto annotate > dimensions > attach type> on entities or center > left click on entity > mid button

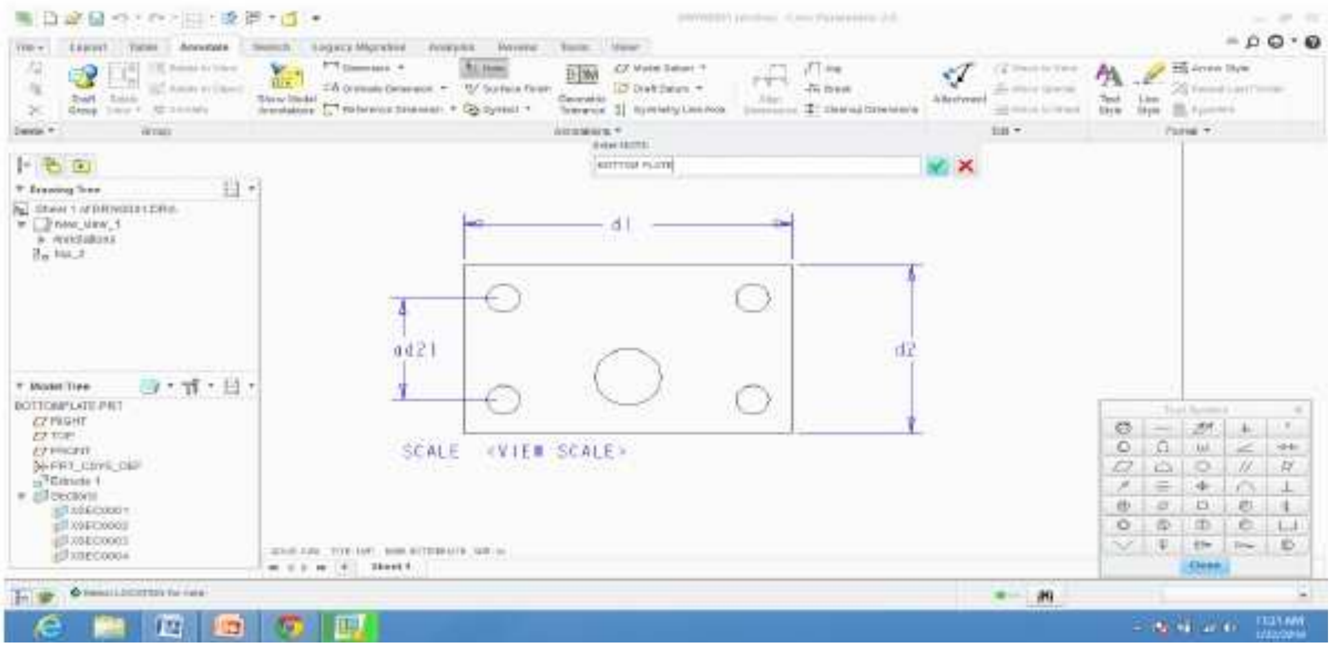
For diameter: 2 times left click on entity > mid button



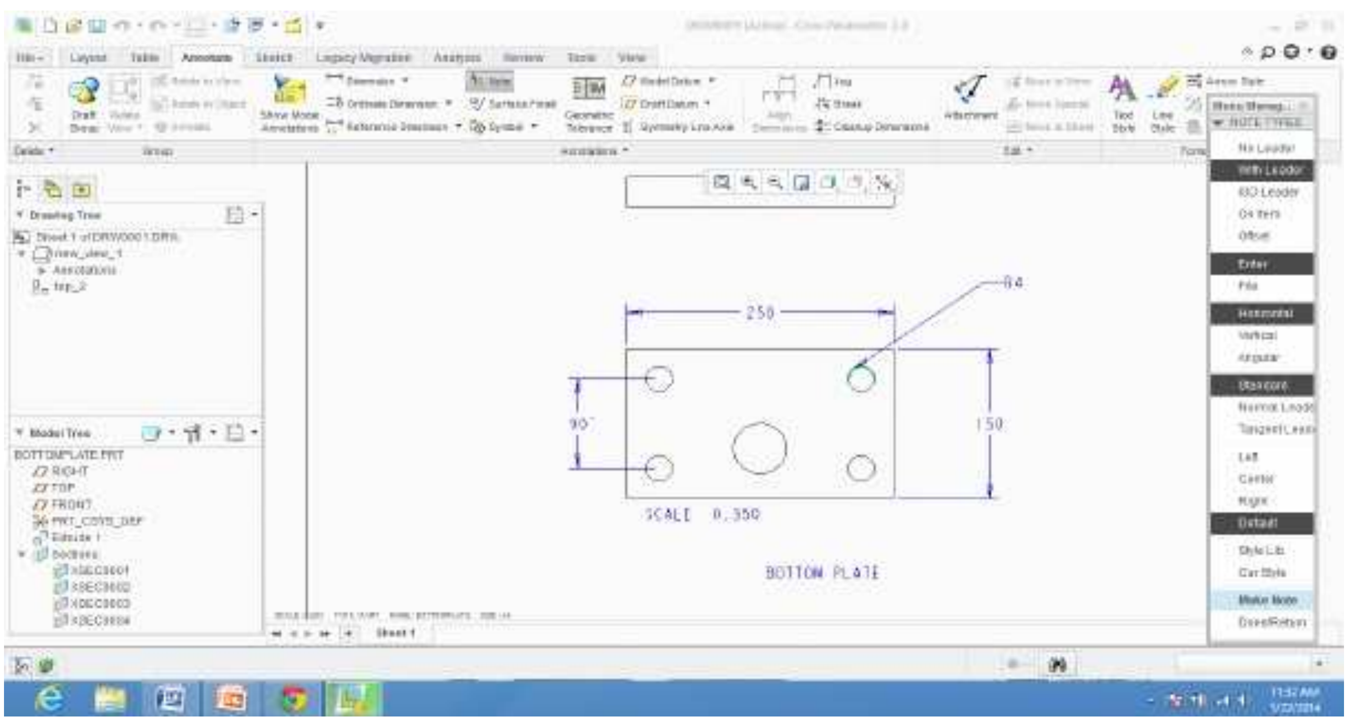


Step 6: Text

Goto make note > no leader > make note > select point > write text note > enter



Goto make note > with leader > make note > attached type >entity > ok > write text > done



Step 7: Creating bill of materials table

a> Select table of (4*2) > left click on cell of table > rt click > change width and height if required

b> Double click on (1*1) cell and write index , similarly (1*2) cell and write assembly member name , click on (1*3) and write quantity , click on (1*4) cell and write material name

c> Click on (2*1) cell > goto select table > select entire row > rt click on selected row > add repeat region > double click on cells

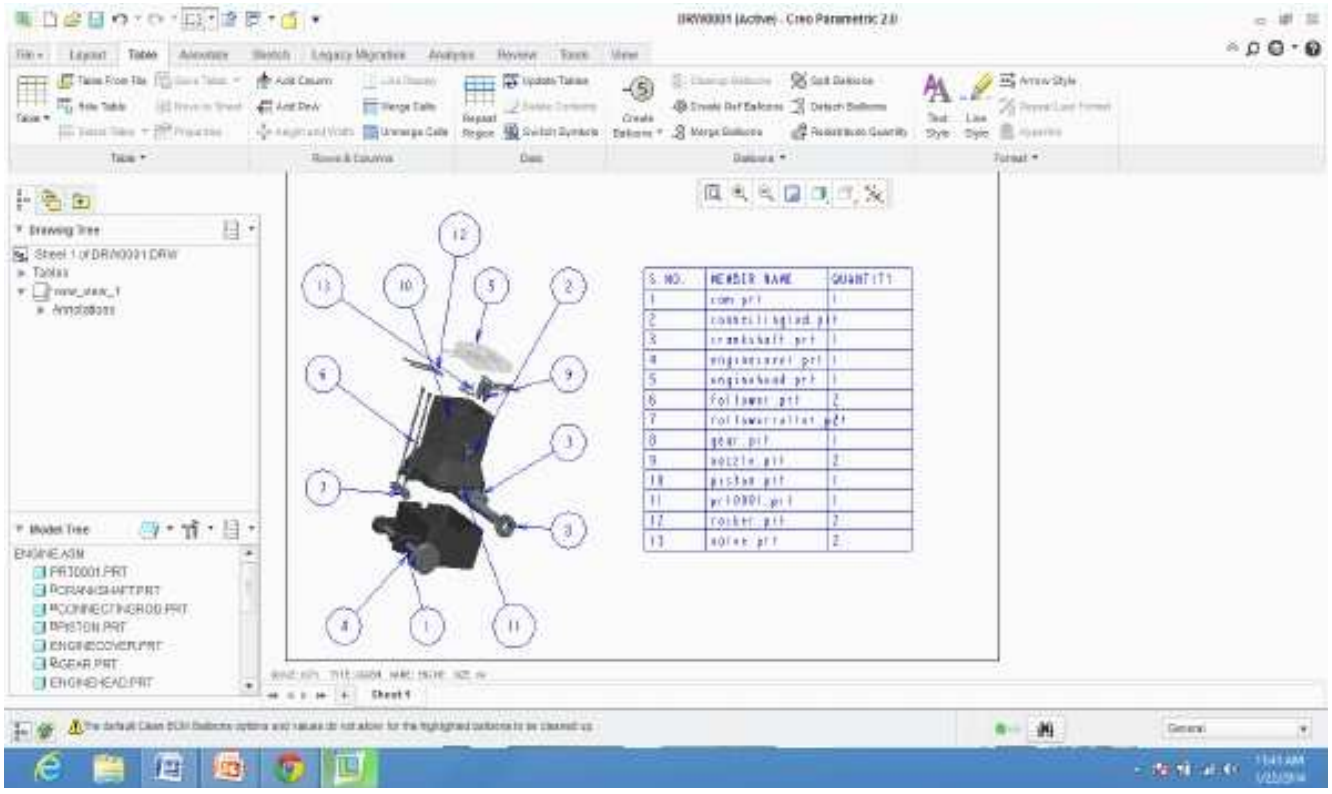
S.NO.	COMPONENT NAME	QUANTITY	MATERIAL NAME
RPT >INDEX	ASM>MBR> PTC_COMMON_NAME	RPT >QTY	ASM>MBR> PTC_MATERIAL> PTC_MATERIAL_NAME

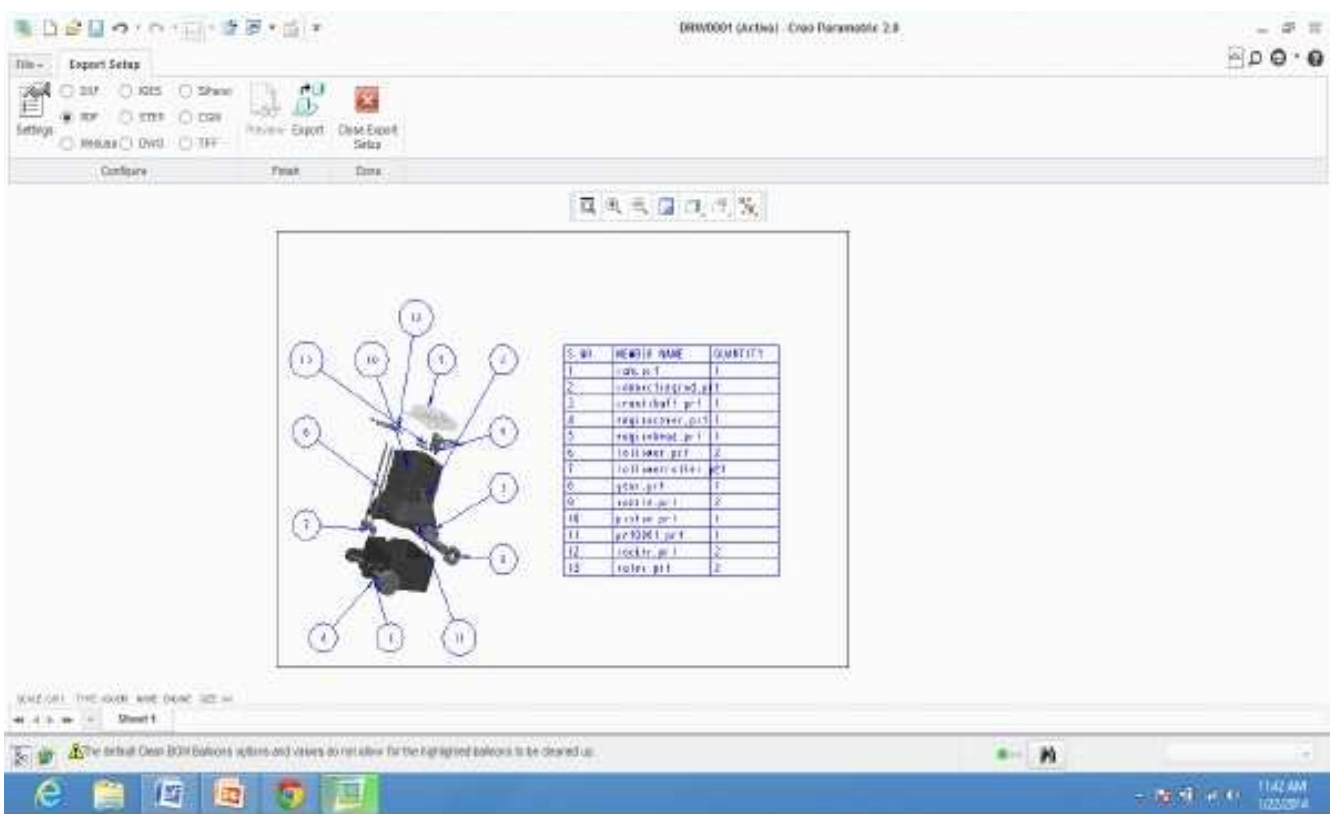
d> Goto repeat region > attributes > select entire 2nd row > no duplicates > done

e> Goto create balloon > all

Step 8: Publishing drafting sheet

Goto file > save as > export > set the format (pdf) > export


DRW0001 (Active) - Creo Parametric 2.0
Drawing Tree
Model Tree
ENGINE.ASM
Sheet 1


DRW0001 (Active) - Creo Parametric 2.0
Export Setup
Sheet 1

CHAPTER 17
ANIMATING ASSEMBLIES

For better presentation of CAD models CREO PARAMETRIC 2.0 provides animation feature .

EXPLODING/IMPLODING @TIME

1 .Click on application – select animation

2 . Click on new animation – select snap shot

A dialog box – define animation pops up

Here name your animation

3 . On right hand top side – click on body definition

A dialog box – bodies. Pops up

Here select one part per body

4 . On left hand top side – click on key frame sequences

A new dialogue box opens

a. Here you can assign a new name
b. In reference body – keep it ground only
c. Click on edit or create a snapshot
d. A new dialog box open-drag

   In the snapshot option take different snap of the model

   In the advanced drag option assign different exploded view to your model and than take snaps .(translation and rotation both)

e. When you click on ok then close you will be guided back to the previous dialogue box

   Here, click on reverse and then ok

5. Double click on the time line to increase the duration of animation .

Animation time domain dialog box is opened

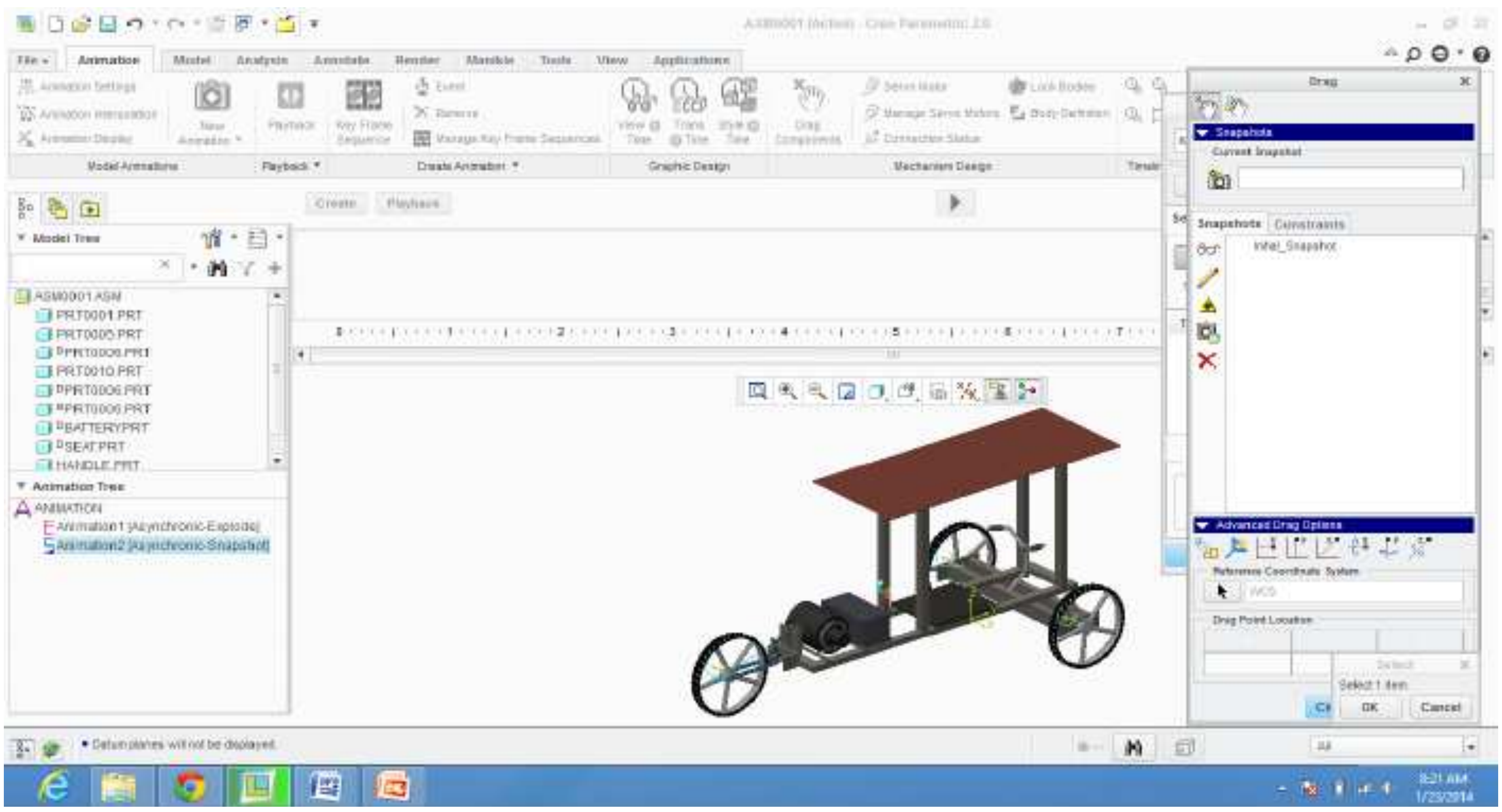
Start time-0.0000

End time-20.0000

Apply then ok

6. Click on generate to see the clip . Manage snapshots w.r.t. time on timeline .

7. Click on play back to save your clip . Click on capture to get MPEG file .



VIEW@TIME

To create this animation you need to first create various customized orientation in assembly module .

1 .Click on application – select animation

2 . Click on new animation – select snap shot

A dialog box – define animation pops up

Here name your animation

3 . On right hand top side – click on body definition

A dialog box – bodies. Pops up

Here select one part per body

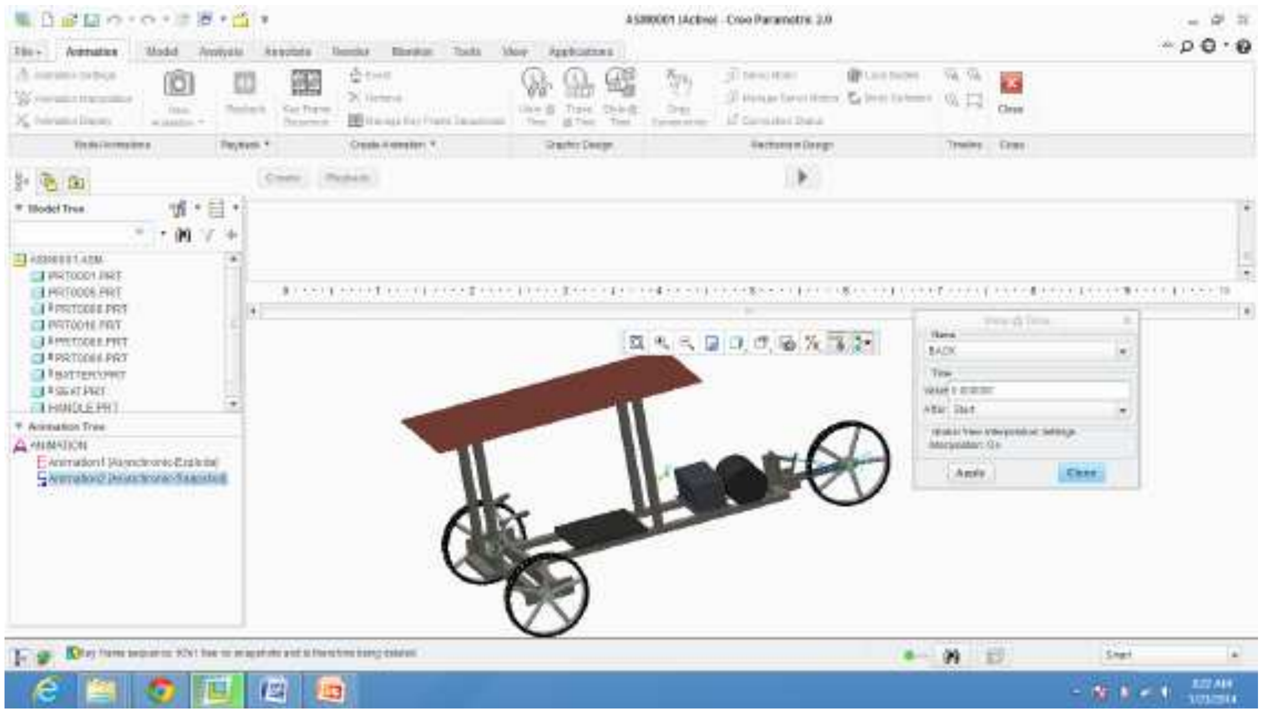
4. Click on VIEW@TIME

5. Set time of clip by double clicking on timeline .

Set orientation w.r.t. time .

Click on done

Check to generate



TRANS@TIME

1 .Click on application – <u>select animation</u>

2 . Click on new animation – <u>select snap shot</u>

A dialog box – <u>define animation</u> pops up

Here name your animation

3 . On right hand top side – click on <u>body definition</u>

A dialog box – <u>bodies.</u> Pops up

Here select <u>one part per body</u>

5. Click on TRANSPARENCY@TIME

Click on cursor icon to select the object to assign transparency

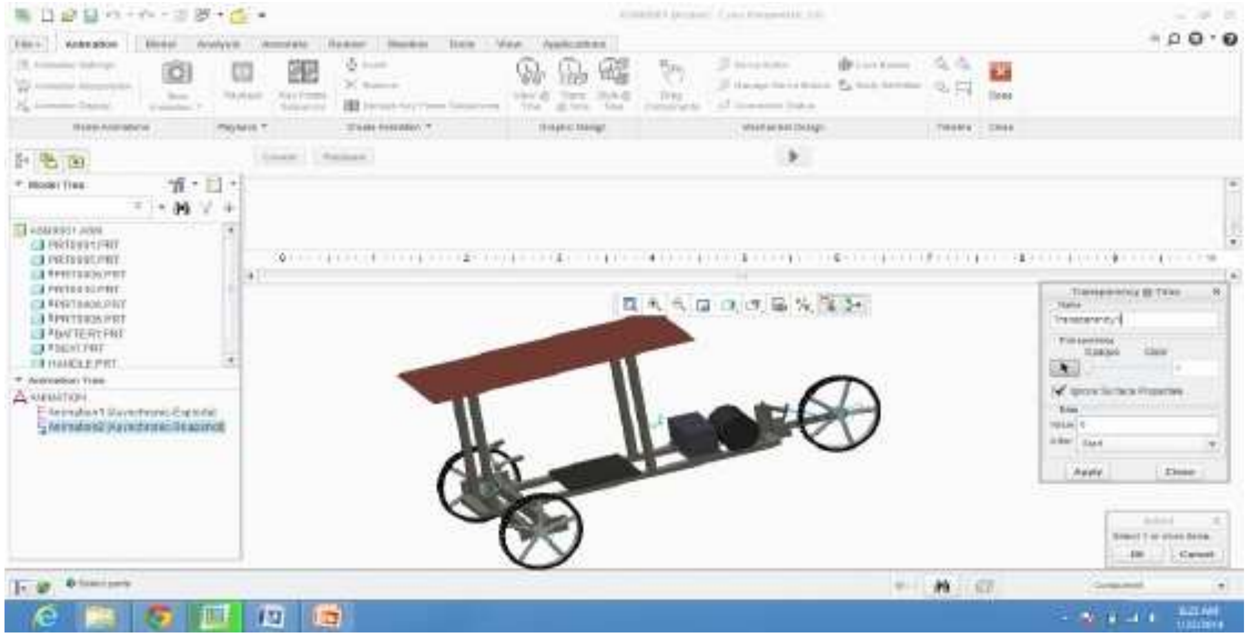
Click ok

Then assign transparency

Assign time

Repeat above 2 steps

Check to generate.



CHAPTER 18
APPLYING MECHANISM TO ASSEMBLIES

When there is no relative motion between components constraints are applied to restrict degree of freedom .

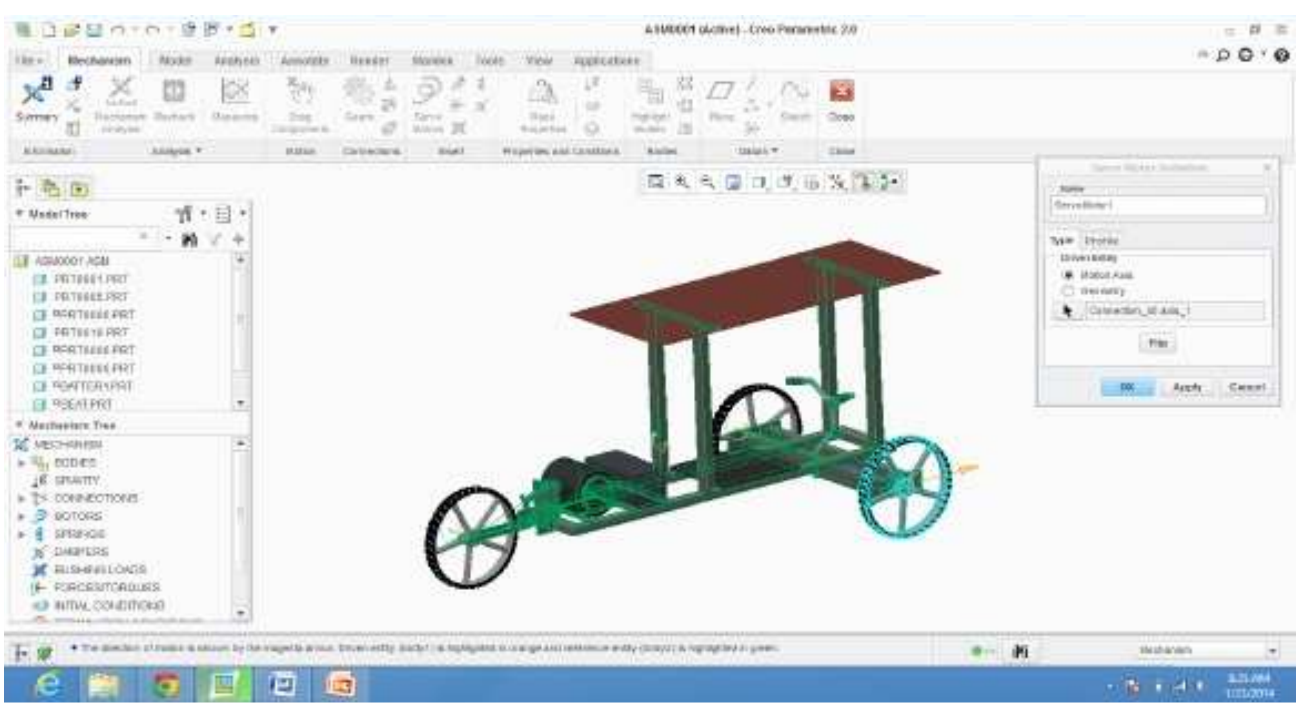
To create relative motion between components we apply connections . In mechanism there is relative motion between components .

For translation and rotational motion we will use **GENERAL** connection . It has 3 translational and 3 rotational degrees of freedom .

To generate **relative motion** between components of assemblies apply **GENERAL** connection.

DEFINE A SERVO MOTOR

A. Servo motor
   - Choose servo motor – servo motor definition dialogue box pops up
     You can see that a arrow icon is visible at every connection
   - Name – you can name your motor
   - Type – select motion axis
   - Click on cursor icon then choose the required connection to which motor is required
   - Profile – in specification change the position motion axis to velocity motion axis
   - In magnitude – constant A=



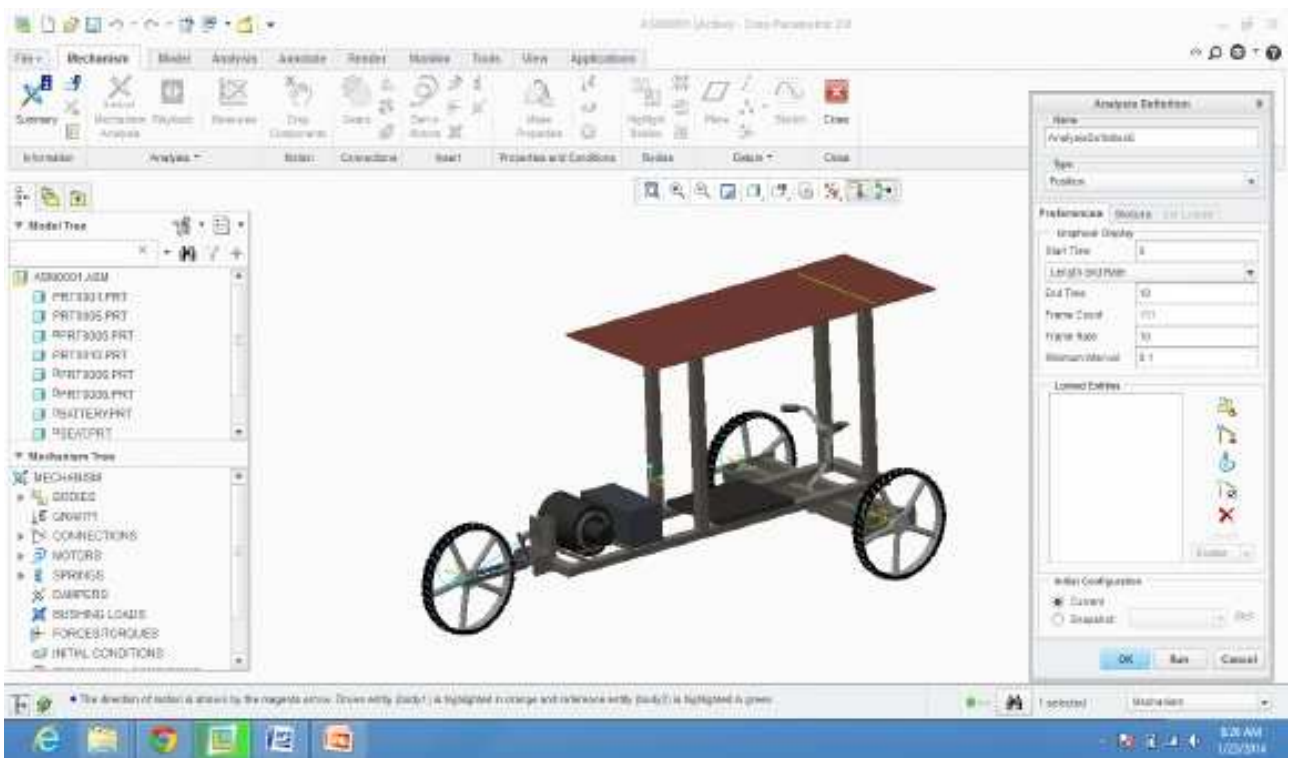
B. Mechanism analysis

- Analysis definition dialogue box pops up
- Type – position
- Preferences – start time=0

  End time=20

- Motors – servomotor1- Start to 10

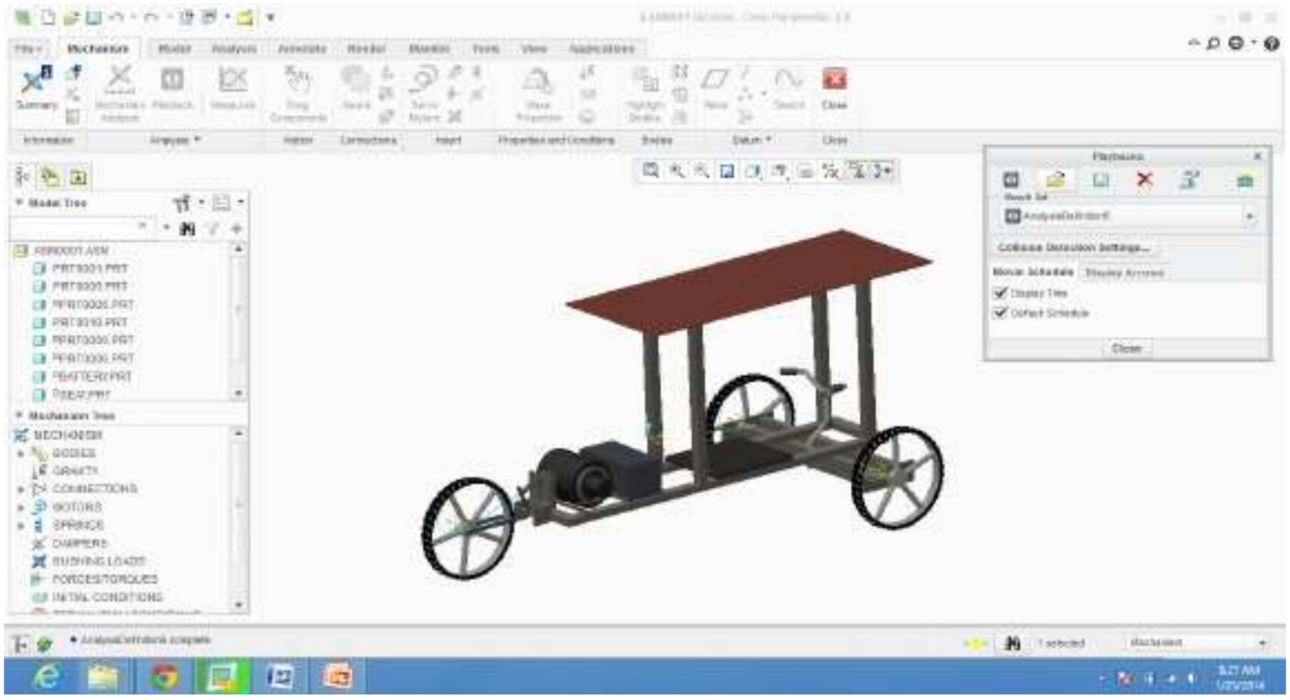
  Servomotor2-11 to end



C. Playback

- Play backs dialogue box pops up
- Result set – chose the analysis definition you wish to capture
- On top click on select play current result set
- Animate dialogue box
- Click on play to see your clip
- Click on capture to save your clip – capture dialogue box pops up
- Click on browse to set the directory where you wish to save your clip

➢ Click on ok and your film starts to play while getting captured



DEFINE A BELT CONNECTION

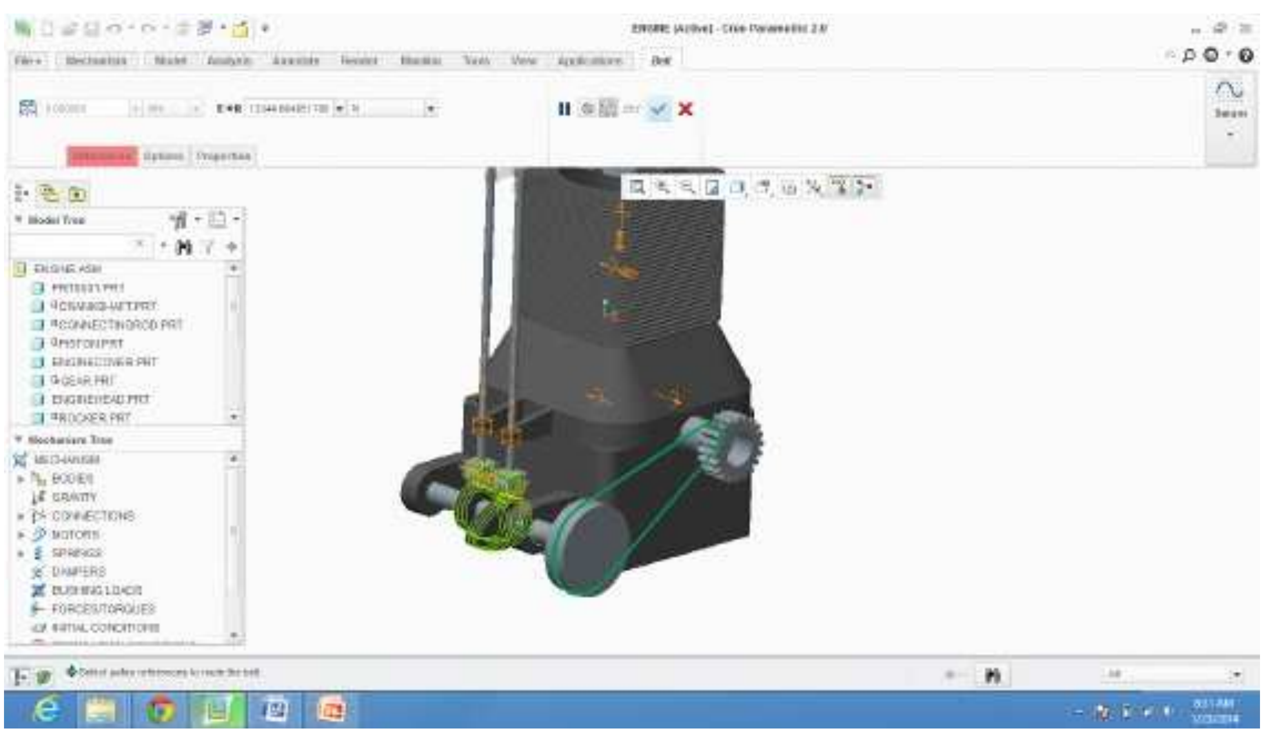
1 . Belts

➢ Click on belt icon to enter into belt defining environment .

➢ Go to references

Pulleys – select the two required pulleys by using ctrl button on keyboard . Now you can see that a belt has formed over pulleys .

Now repeat steps A,B,C .



DEFINE A GEAR PAIR CONNECTION

1. Click on GEARS icon for defining a gear pair.
2. Gear pair definition dialogue box pops up
   a. Name – Name your gear pair
   b. Type – you can assign the type of gear pair you desire
   c. Gear 1 option-

      Motion axis- click on cursor icon to select a motion axis
      Body – pinion-
      Ground-
      Pitch circle – diameter -10mm

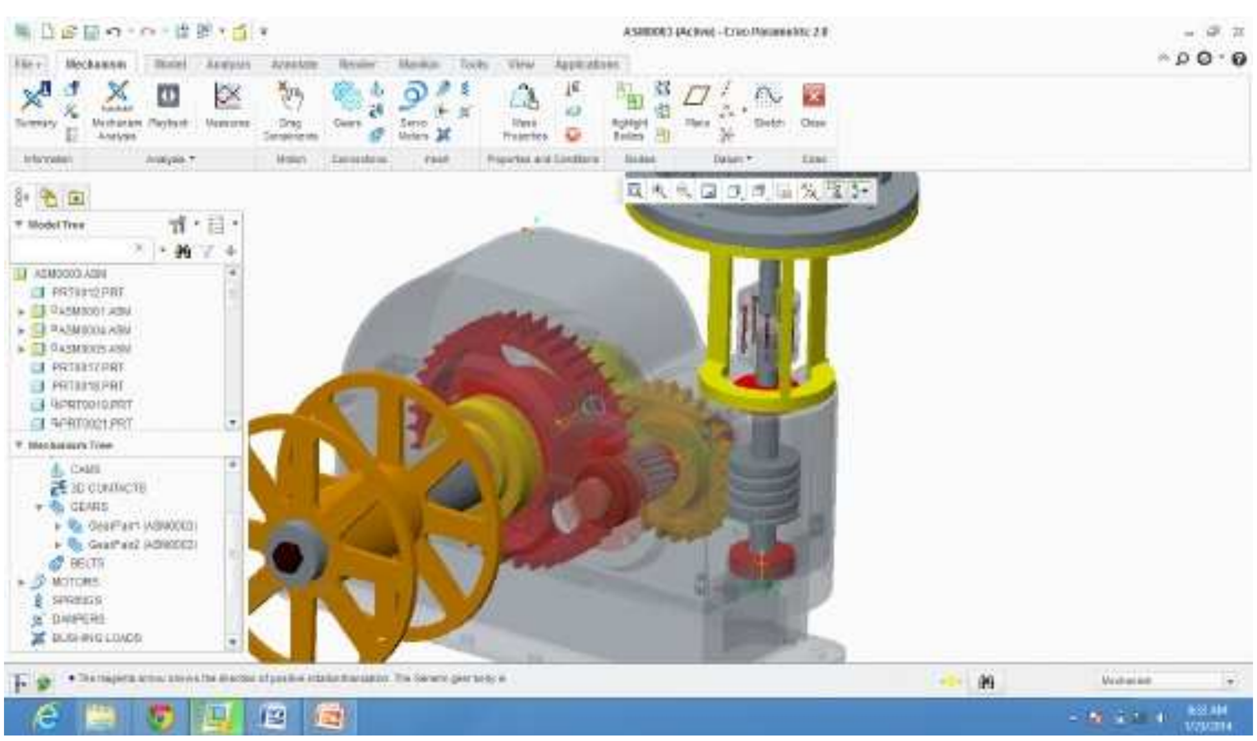
   d. Gear 2 option-

      Motion axis – click on cursor icon to select a motion axis
      Body – Pinion-
      Ground-
      Pitch circle – diameter – 10mm

   e. Pitch circle diameter

      D1-
      D2-

   Repeat steps A,B,C



DEFINE A CAM FOLLOWER CONNECTION

1. Click on CAM icon for definition a cam follower connection
2. Cam follower connection definition dialogue box pops up
    - a> Name – you can cam-follower connection
    - b> Cam 1 option -

        Surfaces / curves – click on cursor icon then select the required

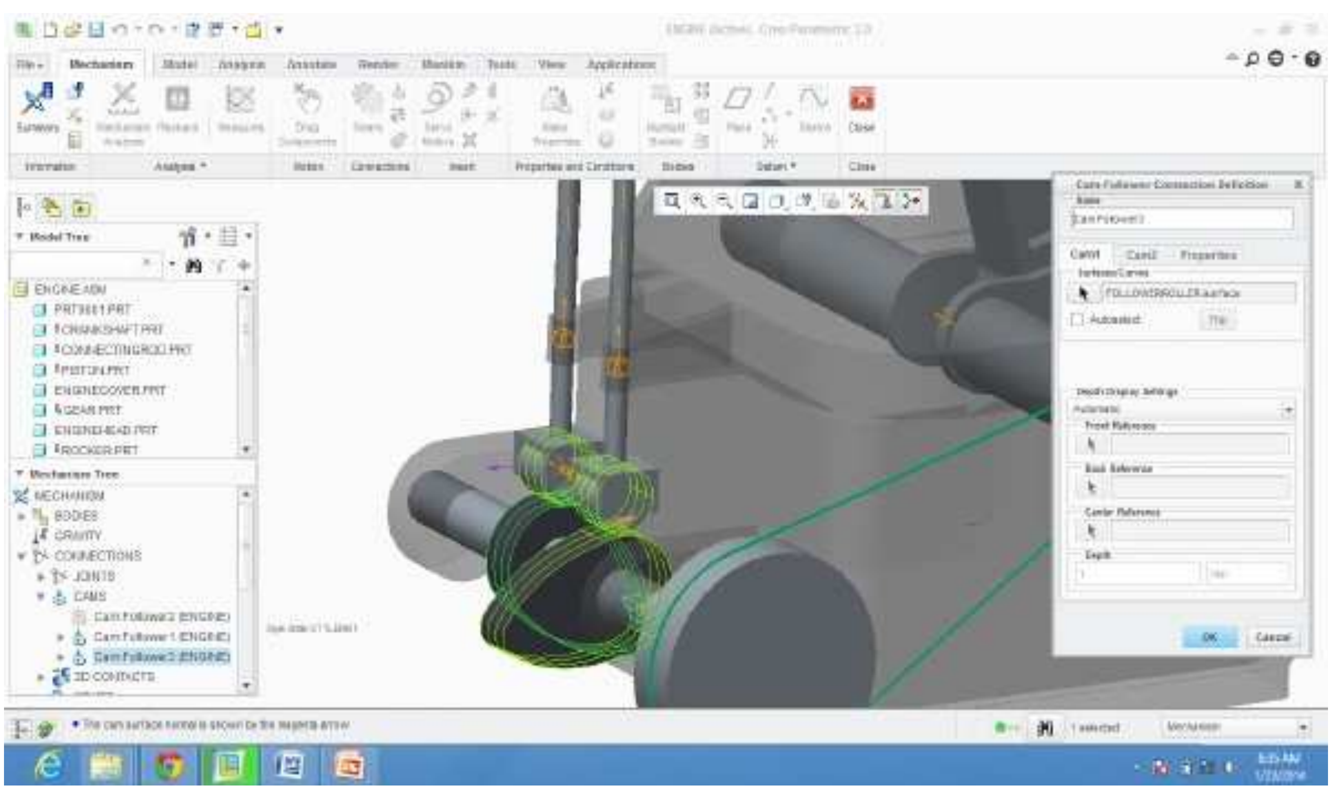
        Surfaces or curves for cam

    - c> Cam 2 option –

        Surfaces/curves – click on cursor icon then select the required surfaces or curves for follower

    - d> Properties – don't make any changes

Repeat steps A,B,C



CHAPTER 19
CREATING SHEET METAL WORK

Step 1: How to invoke a sheet metal ?

a> Select new – new dialogue box pops up

Type	Part
Sub type	Sheet metal
New file option	Mmns_part_sheetmetal

Step 2: Create primary wall – use either

a> Extrude
b> Revolve
c> Planar
d> Boundary blend
e> Sweep
f> Swept blend
g> Rotational bend

Prefer using planar

Select planar and sketch the required primary wall.

Select top left hand side box where you can define its thickness.

Step 3: creating secondary wall – use either

a> Flat
b> Flange

Creating a secondary wall using flat wall

- Click on flat wall icon to enter into flat wall creation environment
- Select the required edge of the primary wall from which you need to attch your flat wall. Creo automatically creates bend between edges .
- Using the user predefined wall shape box , on the top left hand side corner , you can
- Define the shape of your secondary wall.
- From the wall angle box you can define the angle of wall
- Add bends to the attachment edges – you can change the bend rdius of attached radius
- Shape panel – you can sketch the wall as per your requirement
- Offset panel- you can offset wall w.r.t attachment edge
- Relief panel – define the appropriate relief for the wall
- No relief – no relief is applied
- Rip – the part is sheared to allow the wall to bend
- Stretch – a region of the part is stretched to simulate the deformation that occurs
- Rectangular – create a rectangular relief using a specified thickness and depth

- Obround – Create a rounded end relief using a specified thickness and depth .

  ➢ Bend allowance panel –

    1. K factor – The location of neutral line (t/T)
       1/3 when R<2t , and as ½ when R>2t

    2. Y factor –
    3. By bend table

Bend allowance comes from the fact that when sheet metal is bent , the inside surface of the bend is suppressed and the outer surface of bend is stretched

<u>Creating a secondary wall using flange wall</u>

Click on flange wall icon to enter into flat wall creation environment

Select the required edge of primary wall from which you need to attch your flange wall. Bend is created between edges .

- User predefined wall shape box , on top left hand side corner .you can define the shape of your secondry wall.
- Trim and extent box you can difine the extension of wall.
- Add bends to the attachment edges-you can change the bend radius of attached radius.

Other panel option are similar or flat wall,except miter cut.

>miter cut-the edge created between the walls,automatically created between the walls,do not mess with it.
>Relief-a.bend relief –select the type of relief suited best.
c.corner relief –select the type of relief suited best.

Step 4.create an edge rip

General,a rip is a zero-volume cut that is created on a sheet metal part. If your part is designed as a continuous piece of material, it cannot be unbent without ripping the sheet. .Create a rip feature before ending.
Edge rip – select an edge of a surface . Creo creates a saw cut along the edge.
Sketched rip – select the surface on which to create the rip and then sketch the rip section . Creo creates a saw cut along the sketched rip line

Surface rip – Select a surface and rip the geometry out . This option effectively cuts out the selected surface and so actually removes volume .

Rip connect – create a rip connect between two points .

Step 5: Define a bend

The bend group you will find the bend option using which you can create bends on a surface .

- On the top left hand side you will find three options for placement of bend line .

  A> Bend material upto bend line .
  B> Bend material on other side of bend line
  C> Bend material on both side of bend line

Choose only the B option

- Use value to define bend angle – lets you define a bend angle
- Bend to end of surface – bend one edge to the opposite edge
- Enter a bend radius value – you can define radius of bend
- Placement panel – define the surface or the bend line . You will get two green boxes displayed on top of selected surfaces , place the boxes on the opposite edges of surface and then define the bend line .
- Bend line panel – In the sketch option you can draw the bend line
- Transition – you can use this option only when you are in bend to end of surface environment . Lets you bend a particular area of the surface keeping the rest undisturbed .

Define an edge bend

Function similar to round tool . Here the thickness is defined as the inner radius of the bend .
Placement panel – Displays all the selected edges. Here you can define bend radius and also whether you wish to change thickness definition from inner radius to outer radius .

Define a planar bend

Create a bend around an axis that is perpendicular to the selected surface and the sketching plane .

Select planar bend from bend group

Menu manager dialogue box pops up . Select angle and click on done

two new dialogue box pops up .

a> Bend options : angle , planar
b> Menu manager

Menu manager dialogue box- select part bend tbl . Click on done/return .

- Now you must set up a sketch plane . Select the desired plane and click on okay
- Now you must select the the sketch view . Lets just keep it default .
- You are now in sketch environment . Draw the section about which you desire to bend your plane .
- Bend side option – click on okay
- Direction – click on okay
- Define bend angle – click on enter a value . Give the angle value . Click on done
- Sel radius - Click on enter a value . Give the radius value
- Direction – click on okay
- Click on preview to see your work . If satisfied click on okay
- Bend order tables

Bend orders are used to document the order , size and direction in which the bend was created during the manufacturing process .

To document a bend order a complete model must be available .

Now click on the bend order from the bend group drop down menu .

You will see that the entire model is flattened

Sequence – you can create your own sequences consisting of similar types of bends .

You can create bend order tables and display them in sheet metal drawings to document the order and dimensions of bend features . Bend order tables can be updated by reviewing the bend sequence . When a bend order table is stored , the file name is <modelname>.bot . A bend order table cannot be treated or edited on a completely unfolded model.

Step 6 – create an unbend feature

Use the unbend feature to unbend any curved surface on the sheet metal part , whether it is a bend feature or a curved wall.

1> Click on the unbend icon available in the unbend group . You will be in the unbend environment . On the top left hand side you will see two arrow icons .
   a> Reference selected automatically – Creo itself chooses which surface to keep fix
   b> Reference selected manually – you can choose which surface to be fixed

2> Once you have selected references select manually click on references.
   a> Bend geometry – select edges or surface which you want to unbend
   b> Fixed geometry – select your preferred surface which you want to keep fixed during unbend.

   Deformation panel – When you have large model with multiple bends of different cross sections then creo finds it difficult to automatically unbend the non-ruled bends because they curve or bend in more than one direction . Those bends which creo cannot unbend is shown in deformation panel .

   <u>Define a cross sectional driven unbend</u>

   This tool lets you unbend wall that are bent or curved in more than one direction
   - Click on cross sectional driven unbend icon available in unbend group

- New dialogue box menu manager will open up
- Click on one by one , select dialogue box pops up , then select the desired bend you require to unbend
- Click on okay and then done
- The next dialogue box has two options
  A. Select curve – lets you select the bend defining curve .
  B. Sketch curve – lets you draw the fix curve during unbend
- Select curve to be kept fixed . Click on done
- Flip dir dialogue box – lets you specify side to be kept fixed . Click on OK

<u>Define a transition unbend</u>

Choose transition unbend from the UNBEND Opt . The unbend (transition type) dialogue box appears .

- The FEATURE REFS menu appears .Pick one or more planes or edges to remain fixed while unbending . The selected entities highlight in magenta . Note that both the green and white sides of a surface must be selected in order for selection to be valid .
- Choose done sel when you are finished , then choose done refs to continue
- The FEATURE REFS menu reappears , with Add again chosen by default . Choose the Trans Areas element then define button
- Pick the transitional surface that needs to be unbent . Again both the green and white sides of a surface must be selected in order for selection to be valid .Adjacent edge surfaces must be selected also . The selected entities highlight in magenta .
- Choose done sel when you are finished ; then Done Refs from the FEATURE REFS menu.
- The feature is now fully defined. Choose OK from the dialogue box . The system creates the TRANSITIION Unbend.

Step 7 . Create a bend back feature

Bend back feature is used to " rebend " a part after an Unbend feature has been created . When you create a bend back feature , you select a fixed plane . You can then bend back all or selected bends to bend back. Any features added to the part in this state are automatically bent when the part is bent . This combination of Unbend and Bend back is often used to create cuts that are difficult to model with the part in the formed position , such as the tab feature . In addition you may find that you can create some shapes with this technique that cannot be modeled in any other way.

Step 8 . create a form features

Form features are used to create shapes which are stamped into sheet metal . You can control the shape of the form as either a Punch or a Die . The flatten form feature is used to flatten the form feature for display in the part's flat state . A form feature is typically used to create shapes which deform the surfaces of sheet metal part , such as an extruded hole or a stamped pocket . Form features get their shape definition from the surfaces of an existing part , known as reference part . There are 2 types of form features available in SHEET METAL : Punch and Die . Their use depends upon whether your reference geometry is in the form of the punch (the positive shape), or the die (the negative shape).

- Create a die form
  When you click on die form . Menu manager dialogue box options pops up . Here you must choose between copy and reference
  a> Reference : The feature is actually tied to the refrence part . When the reference part changes , so does the form feature , because they are associated .
  b> Copy : The geometry is copied into the part from the reference part . The form feature will then behave just like any other feature .

After clicking on reference . Click on done . Now open dialogue box pops up . Here you must call your desired shape from the Creo memory . The path will be :

C:\ Program Files\ Creo 2.0 \ Common Files \ F000\ text \ smt \ punch_models\component placement dialogue box pops up . The box functions similar to assembly .

Here assemble the forms required surface to model's surface and fully constraint the assembly . Use coincident and fix.

Define bond plane : select the surrounding surface to the die geometry .

Define seed surface : select any surface of the form . * top most surface is recommended *

Excluded surface : select the surface you don't want in form .

Click ok

- Create a punch form
  Click on the punch form icon . You will enter into punch form environment .
  On the top left hand corner you will find punch model box , where you can choose from different kind of punch .

  There are three different methods to place your punch form .

  a> Place using interface – a warning box pops up . click on yes
  You will see that cursor is holding the Cartesian system of punch , you must coincide with models cartesian system . So this option is used only when you want your punch to be placed at the cart_sys of model .

  b> Place manually – placing of punch using assembly placement methodology .

  c> Place using coordinate system – here you select the surface on which you want to punch .

Placement panel :

a> Reference - select the surface on which you want to punch
b> Placement direction – you may change the direction of punch
c> Type – keep it linear
d> Offset reference – place the green boxes at the edges of the model
e> Add rotation – you rotate your punch at a degree