Introduction to CATIA V5

Release 19

(A Hands-On Tutorial Approach)
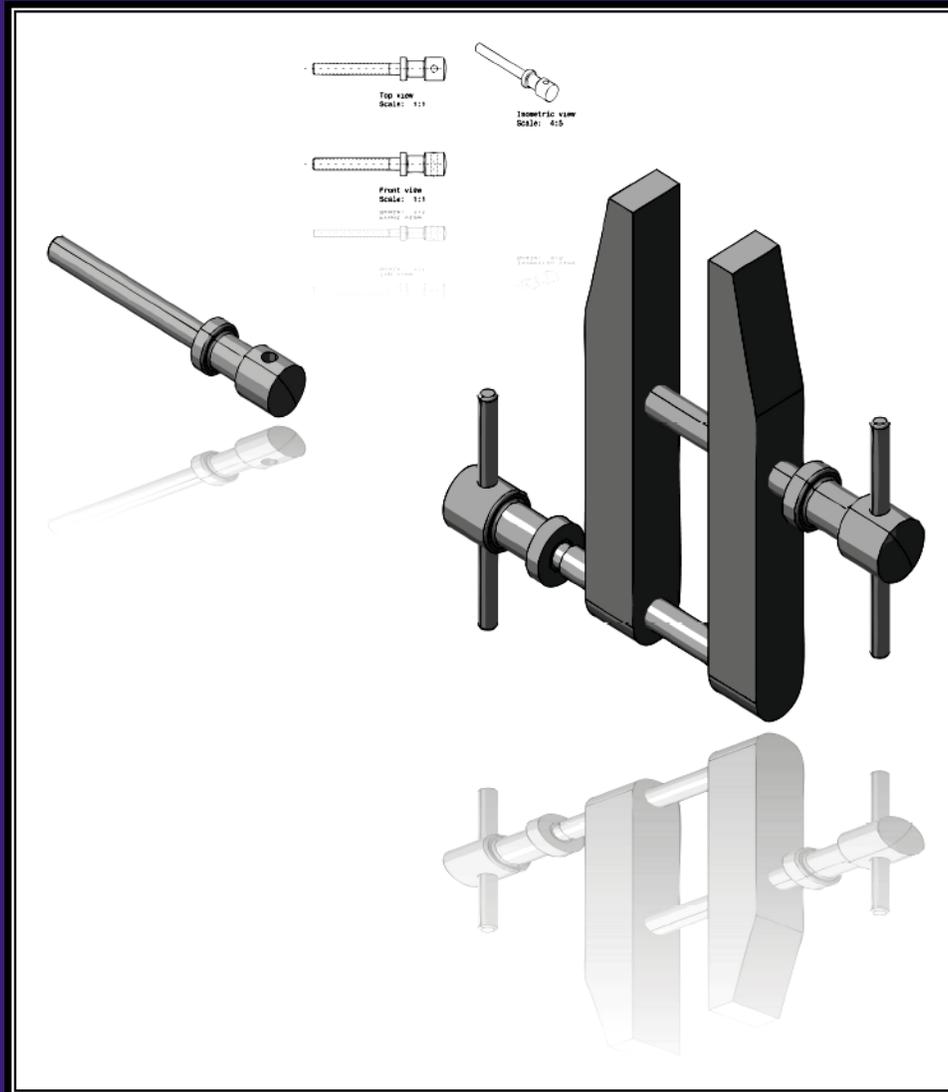
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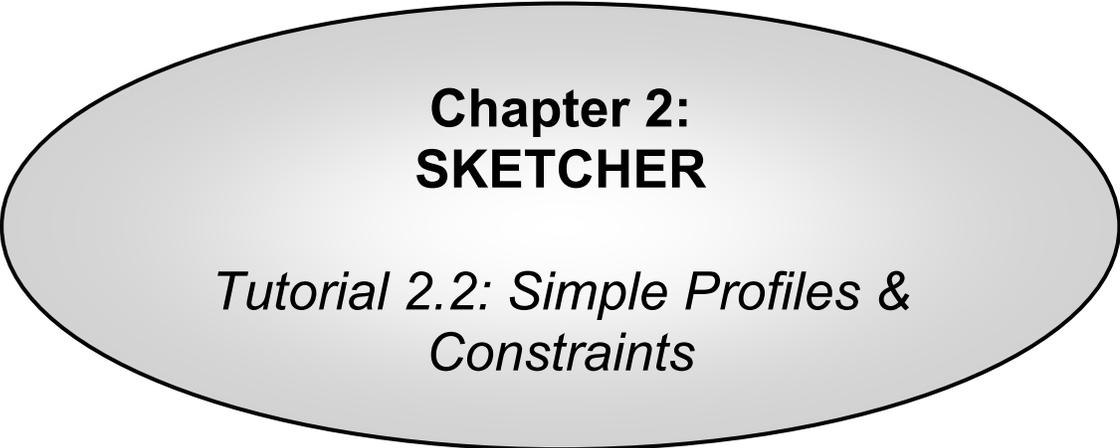
Chapter 2: SKETCHER

Tutorial 2.2: Simple Profiles & Constraints

Featured Topics & Commands

Prerequisite Knowledge & Commands

- Entering workbenches
- Entering and exiting the *Sketcher* workbench
- Simple Pads
- Work modes (Sketch tools toolbar)

Profile toolbar

The *Profile* toolbar contains 2D geometry commands. These geometries range from the very simple (point, rectangle, etc...) to the very complex (splines, conics, etc...). The *Profile* toolbar contains many sub-toolbars. Most of these sub-toolbars contain different options for creating the same geometry. For example, you can create a simple line, a line defined by two tangent points, or a line that is perpendicular to a surface.
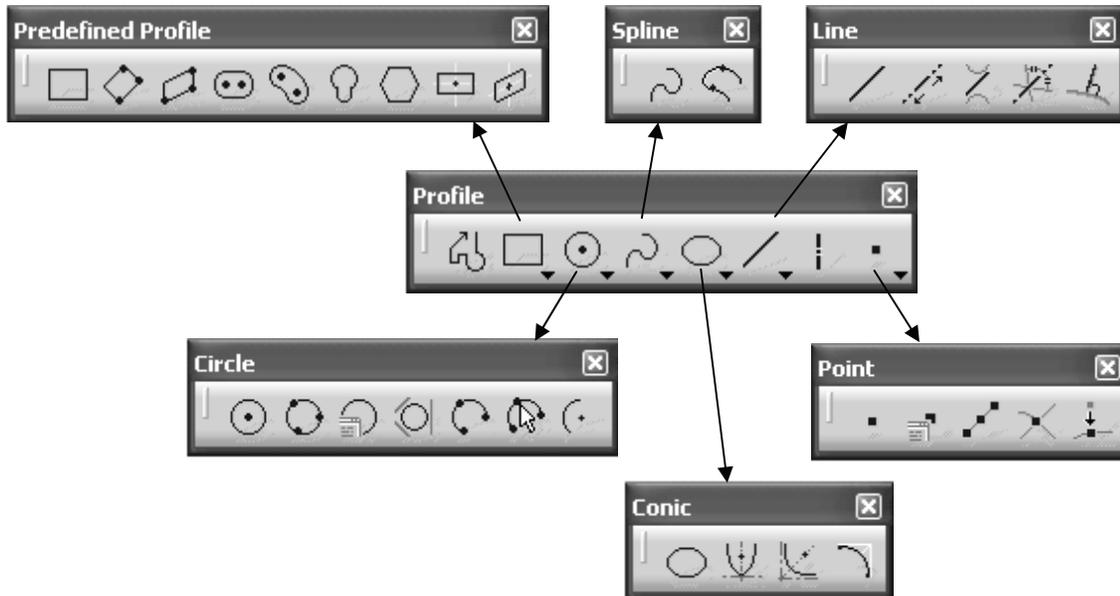
Profile toolbar

Reading from left to right, the *Profile* toolbar contain the following commands.

- Profile: This command allows you to create a continuous set of lines and arcs connected together.
- Rectangle / *Predefined Profile* toolbar: The default top command is *rectangle*. Stacked underneath are several different commands used to create predefined geometries.
- Circle / *Circle* toolbar: The default top command is *circle*.   Stacked underneath are several different options for creating circles and arcs.
- Spline / *Spline* toolbar: The default top command is *spline* which is a curved line created by connecting a series of points.
- Ellipse / *Conic* toolbar: The default top command is *ellipse*.   Stacked underneath are commands to create different conic shapes such as a hyperbola.
- Line / *Line* toolbar: The default top command is *line*. Stacked underneath are several different options for creating lines.

- Axis: An axis is used in conjunction with commands like mirror and shaft (revolve). It defines symmetry. It is a construction element so it does not become a physical part of your feature.
- Point / *Point* toolbar: The default top command is *point*. Stacked underneath are several different options for creating points.

Predefined Profile toolbar

Predefined profiles are frequently used geometries. CATIA makes these profiles available for easy creation which speeds up drawing time. Reading from left to right, the *Predefined Profile* toolbar contains the following commands.

- Rectangle: The *rectangle* is defined by two corner points. The sides of the rectangle are always horizontal and vertical.
- Oriented Rectangle: The *oriented rectangle* is defined by three corner points. This allows you to create a rectangle whose sides are at an angle to the horizontal.
- Parallelogram: The *parallelogram* is defined by three corner points.
- Elongated Hole: The elongated hole or slot is defined by two points and a radius.
- Cylindrical Elongated Hole: The *cylindrical elongated hole* is defined by a cylindrical radius, two points and a radius.
- Keyhole Profile: The *keyhole profile* is defined by two center points and two radii.
- Hexagon: The *hexagon* is defined by a center point and the radius of an inscribed circle.
- Centered Rectangle: The *centered rectangle* is defined by a center point and a corner point.
- Centered Parallelogram: The *centered parallelogram* is defined by a center point (defined by two intersecting lines) and a corner point.

Circle toolbar

The *Circle* toolbar contains several different ways of creating circles and arcs. Reading from left to right, the *Circle* toolbar contains the following commands.

- Circle: A *circle* is defined by a center point and a radius.
- Three Point Circle: The *three point circle* command allows you to create a circle using three circumferential points.
- Circle Using Coordinates: The *circle using coordinates* command allows you to create a circle by entering the coordinates for the center point and radius in a *Circle Definition* window.

- Tri-Tangent Circle: The *tri-tangent circle* command allows you to create a circle whose circumference is tangent to three chosen lines.
- Three Point Arc: The *three point arc* command allows you to create an arc defined by three circumferential points.
- Three Point Arc Starting With Limits: The *three point arc starting with limits* allows you to create an arc using a start, end, and midpoint.
- Arc: The *arc* command allows you to create an arc defined by a center point, and a circumferential start and end point.

Spline toolbar

Reading from left to right, the *Spline* toolbar contains the following commands.

- Spline: A spline is a curved profile defined by three or more points. The tangency and curvature radius at each point may be specified.
- Connect: The connect command connects two points or profiles with a spline.

Conic toolbar

Reading from left to right, the *Conic* toolbar contains the following commands.

- Ellipse: The ellipse is defined by a center point and major and minor axis points.
- Parabola by Focus: The parabola is defined by a focus, apex and start and end points.
- Hyperbola by Focus: The hyperbola is defined by a focus, center point, apex and start and end points.
- Conic: There are several different methods that can be used to create conic curves. These methods give you a lot of flexibility when creating the above three types of curves.

Line toolbar

The *Line* toolbar contains several different ways of creating lines. Reading from left to right, the *Line* toolbar contains the following commands.

- Line: A line is defined by two points.
- Infinite Line: Creates infinite lines that are horizontal, vertical or defined by two points.
- Bi-Tangent Line: Creates a line whose endpoints are tangent to two other elements.
- Bisecting Line: Creates an infinite line that bisects the angle created by two other lines.

- Line Normal to Curve: This command allows you to create a line that starts anywhere and ends normal or perpendicular to another element.

Point toolbar

The *Point* toolbar contains several different ways of creating points. Reading from left to right, the *Point* toolbar contains the following commands.

- Point by Clicking: Creates a point by clicking the left mouse button.
- Point by using Coordinates: Creates a point at a specified coordinate point.
- Equidistant Points: Creates equidistant points along a predefined path curve.
- Intersection Point: Creates a point at the intersection of two different elements.
- Projection Point: Projects a point of one element onto another.

Constraint toolbar

Constraints can either be dimensional or geometrical. Dimensional constraints are used to constrain the length of an element, the radius or diameter of an arc or circle, and the distance or angle between elements. Geometrical constraints are used to constrain the orientation of one element relative to another. For example, two elements may be constrained to be perpendicular to each other. Other common geometrical constraints include parallel, tangent, coincident, concentric, etc... Reading from left to right:
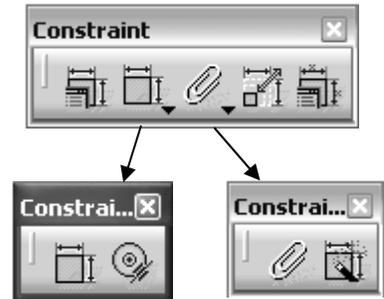
- Constraints Defined in Dialoged Box: Creates geometrical and dimensional constraints between two elements.
- Constraint: Creates dimensional constraints.
  - Contact Constraint: Creates a contact constraint between two elements.
- Fix Together: The fix together command groups individual entities together.
  - Auto Constraint: Automatically creates dimensional constraints.
- Animate Constraint: Animates a dimensional constraint between to limits.
- Edit Multi-Constraint: This command allows you to edit all your sketch constraints in a single window.

Selecting icons

When an icon is selected, it turns orange indicating that it is active. If the icon is activated with a single mouse click, the icon will turn back to blue (deactivated) when the operation is complete. If the icon is activated with a double mouse click, it will remain active until another command is chosen or if the Esc key is hit twice.

Tutorial 2.2 Start: Part Modeled

The part modeled in this tutorial is shown on the right. This part will be created using simple profiles, circles, arcs, lines, and hexagons.   The   geometries   are constrained   to   conform   to   certain dimensional (lengths) and geometrical constraints (tangent, perpendicular, etc...).

Section 1: Creating circles.

(Hint: If you get confused about how to apply the different commands that are used in this tutorial, read the prompt line for additional help.)

1) Open a **New…** part and name your part *Post*.

2) **Save** your drawing as *T2-2.CATPart*.

3) Enter the **Sketcher** on the **zx plane**.
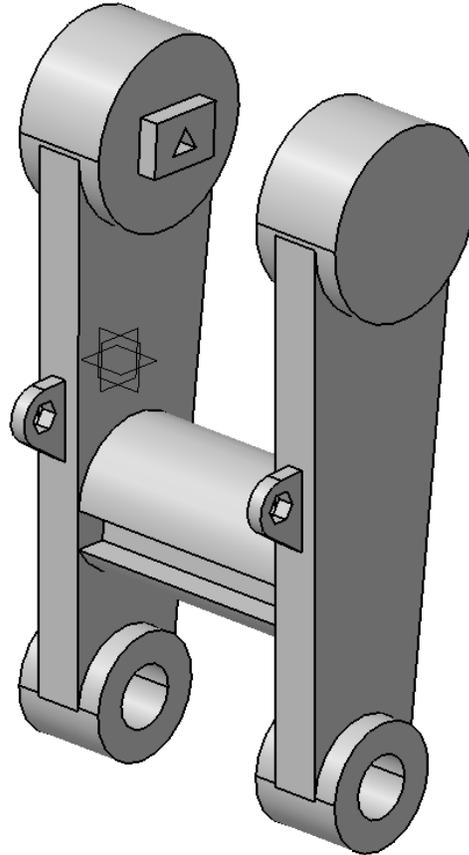
4) Set your grid spacing to be **100 mm** with **10** graduations, activate the *Snap to point*, and activate the *geometrical* and *dimensional* constraints. (**Tools – Options...**)
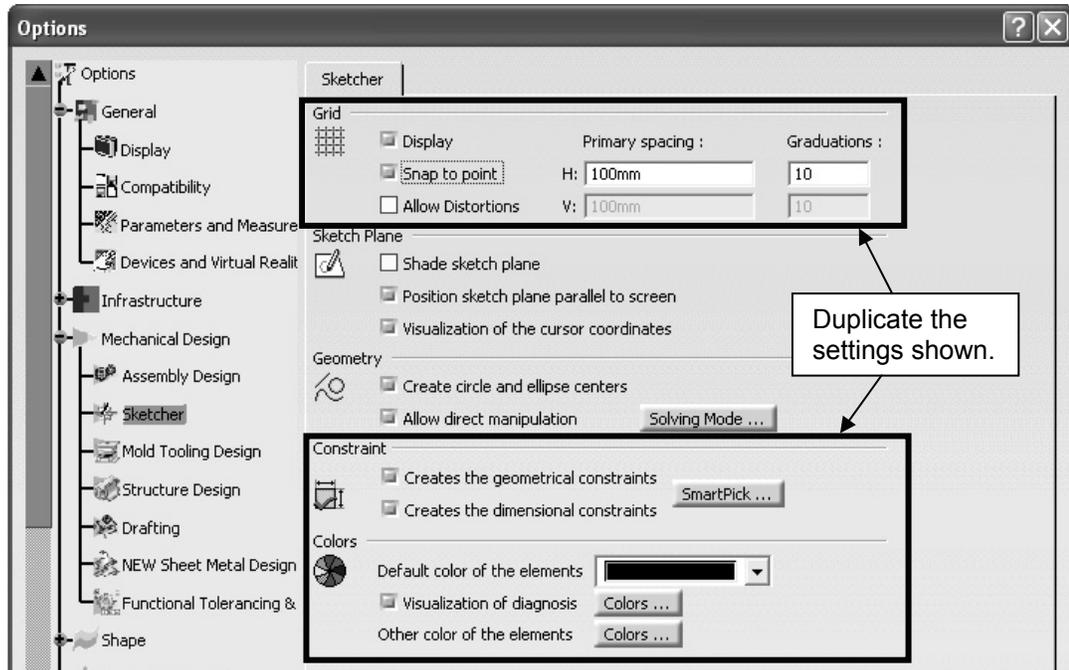
5) Pull out the **Circle** subtoolbar .

6) Double click on the **Circle** ⊙ icon and draw the circles shown.

7) **Exit** the Sketcher ⬆ and **Pad** ⯐ the sketch to *12 mm* on each side (**Mirrored extent**). Notice that the inner circle at the bottom becomes a hole.

Pad Definition [?][X]

First Limit

Type: Dimension ▼

Length: 12mm ⬍

Limit: No selection

Profile/Surface

Selection: Sketch.1 🖉

☐ Thick

Reverse Side

☑ Mirrored extent

Reverse Direction

More>>

● OK   ● Cancel   Preview
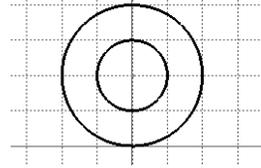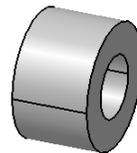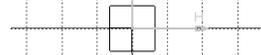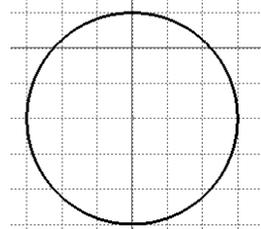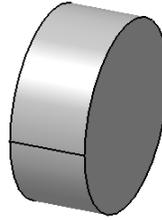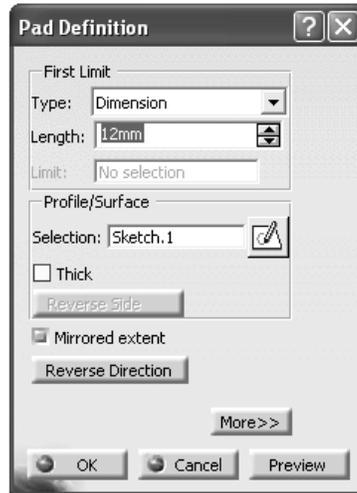
Section 2: Creating dimensional constraints.

1) Expand your specification tree to the sketch level.

2) Edit Sketch.1. To edit a sketch you can double click on the sketch name in the specification tree, or you can *right click* on the name select **Sketch.1 - Edit**. CATIA automatically takes you into the sketcher on the plane used to create Sketch.1.

Post
— xy plane
— yz plane
— zx plane
— PartBody
— Pad.1
— Sketch.1
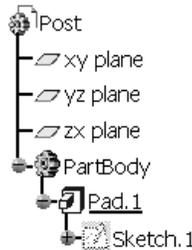
3) Double click on the **Constraints** 🖵 icon.

4) Select the border of the upper circle, pull the dimension out and click your left mouse button to place the dimension. Repeat for the two bottom circles.

5) Select the center point of the upper circle, then the center point of the lower circles, pull the dimension out and click.
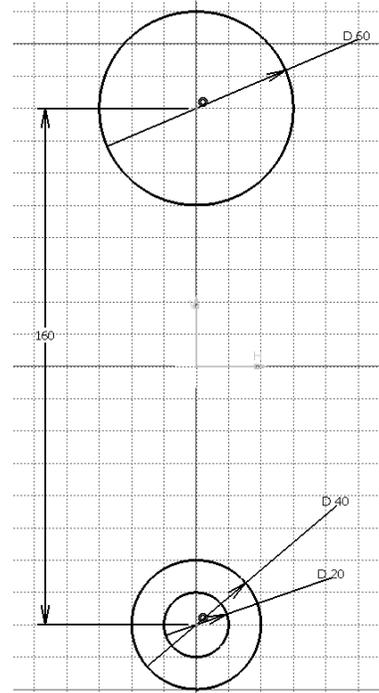
6) Double click on the **D20** dimension. In the *Constraint Definition* window, change the diameter from 20 to **16 mm**.

Constraint Definition

Diameter 16   ☐ Reference

Dimension Diameter

More>>

OK   Cancel
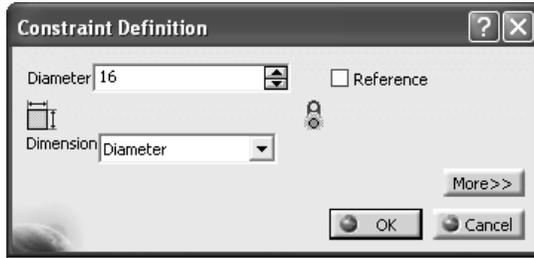
7) In a similar fashion, change the other dimensions to the values shown in the figure.

8) **Exit** the Sketcher and deselect all. Notice that the part automatically updates to the new sketch dimensions.

D48

140

80

D16

D32
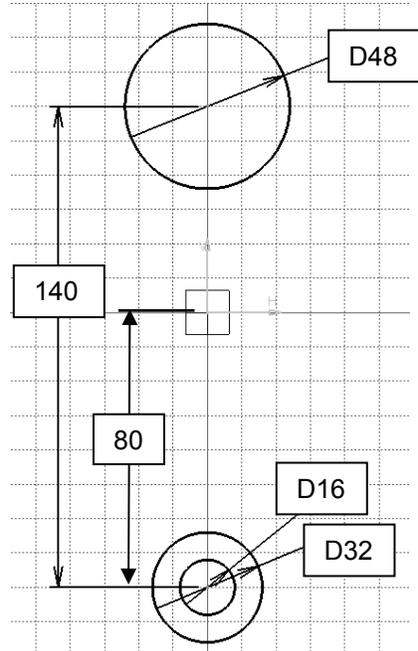
Section 3: Creating lines.

1) Deselect all.

2) Enter the **Sketcher** on the **zx plane**.

3) Deactivate the **Snap to Point** icon.

4) Project the two outer circles of the part onto the sketch plane as Standard elements. Double click on the **Project 3D Elements** icon. This icon is located in the lower half of the right side toolbar area. Select the outer edges of the two cylinders.
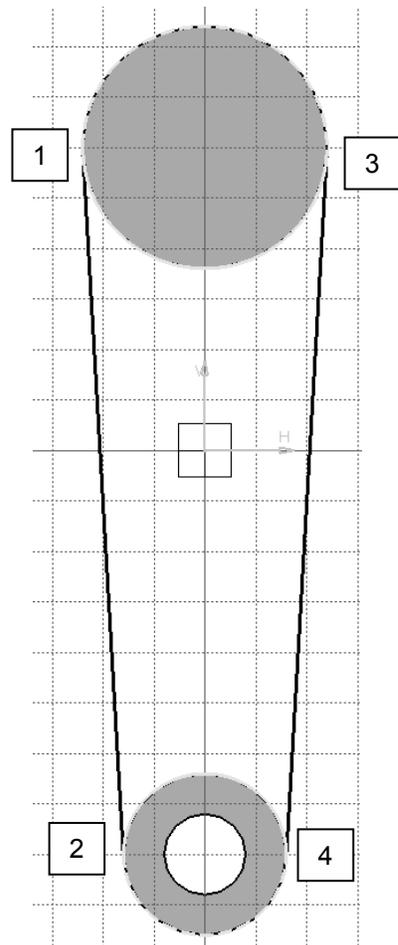
Line

5) Pull out the *line* toolbar .

6) Pull out the *Relimitations* toolbar located in the

Relimitations

Operation toolbar.

1

3

2

4

7) Double click on the **Bi-Tangent Line** icon. Draw two tangent lines by selecting the points, in order, as indicated on the figure.

8) Double click on the **Quick trim** icon. Select the outer portion of the projected circles. Notice that the trimmed projection turns into a construction element (dashed).

9) **Exit** the Sketcher and **Pad** the sketch to *6 mm* on each side (**Mirrored extent**).

Trimmed edge

Projected edge

1

3

Projected edge

2

4

Trimmed edge

Pad Definition

First Limit

Type: Dimension

Length: 6mm

Limit: No selection

Profile/Surface

Selection: Sketch.2

☐ Thick

Reverse Side

◼ Mirrored extent

Reverse Direction

More>>

OK   Cancel   Preview

LIM2   LIM1
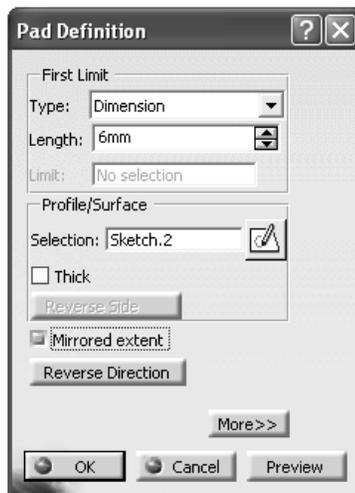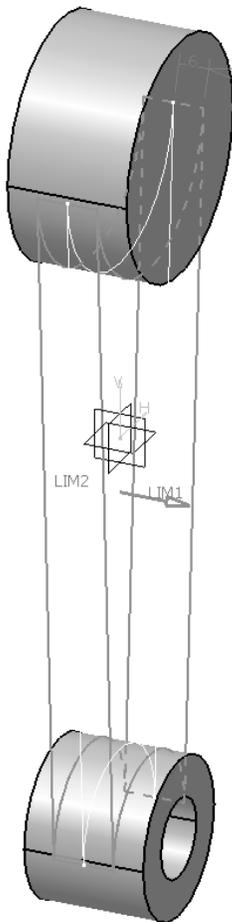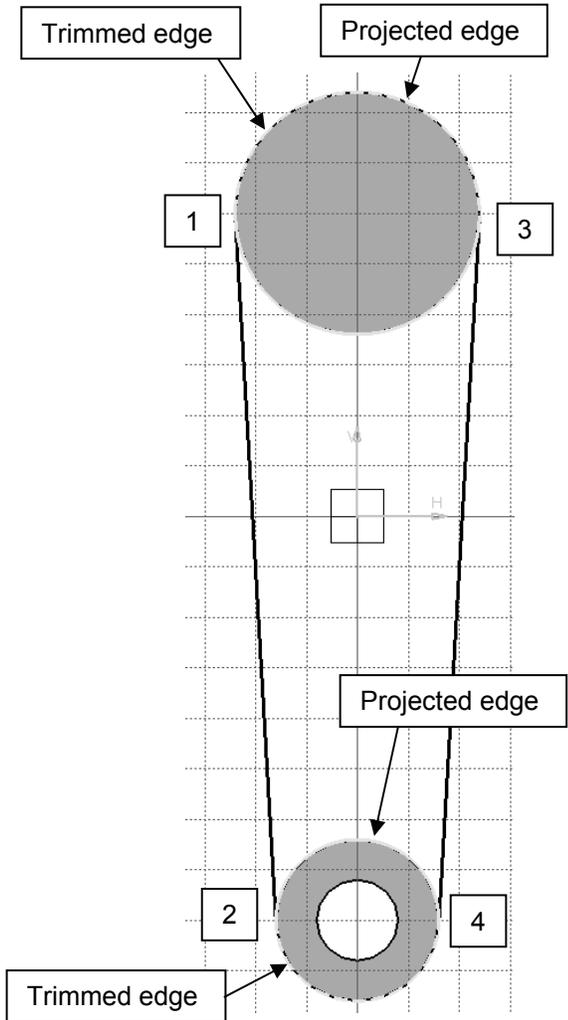
10) **Save** your drawing.

11) Enter the **Sketcher** ⬜ on the **zx plane**.

12) Activate the **Construction/Standard Element** 🔵 icon (it should be orange).

13) Select the **Project 3D Elements** ⬛ icon and then project the left line of the part as shown in the figure. The projected line should be dashed.

14) Activate your **Snap to Point** ⬛ icon.

15) Draw a line that starts at point 1 (see fig.) and ends normal/perpendicular to projected line using the **Line Normal to Curve** ⬛ icon.

16) Deactivate your **Snap to Point** ⬛ icon.

17) Draw a **Line** ⬜ from point 1 to point 2.

18) Draw a line that bisects the previous 2 lines using the **Bisecting Line** ⬛ icon. Read the prompt line for directions.

19) Deselect all.

20) Deactivate the **Construction/Standard Element** 🔵 icon (it should be blue now).

Projected line

Normal line

1

Bisecting line

2
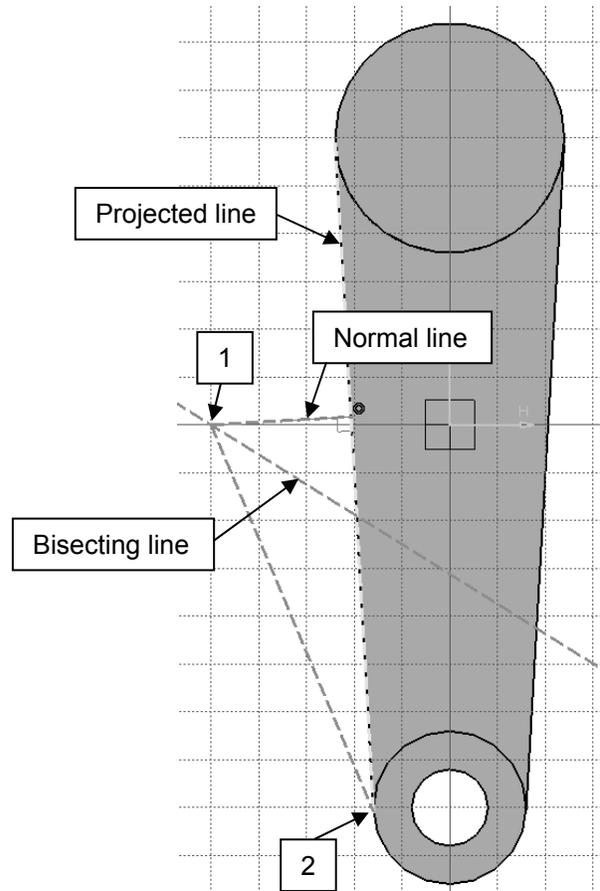
21) Draw a circle that is tangent to the projected line, normal line and bisecting line using the

Tri-Tangent Circle icon.   Read the prompt line for directions.
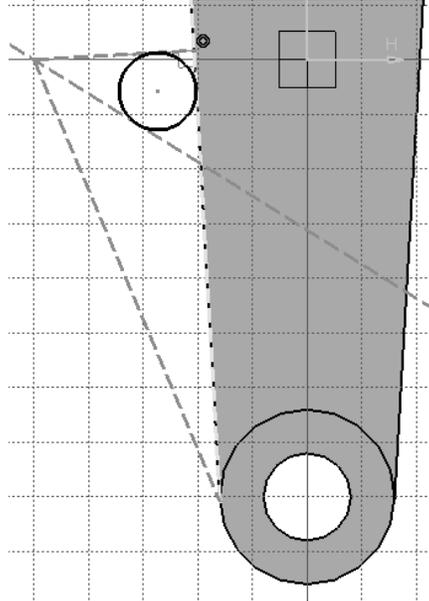
22) Zoom in on the circle.

23) Using **Profile**   , draw the three additional lines shown in the figure. When creating the line that touches the circle, both the construction line and the circle should turn orange before the point is selected.
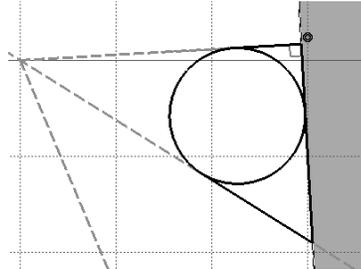
24) Use the **Quick Trim** command to trim off the inside portion of the circle as shown. You will have to apply the quick trim operation twice.
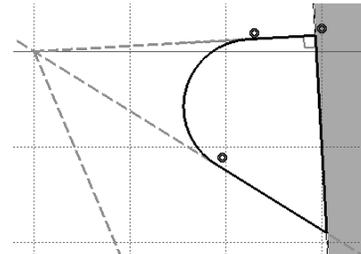
25) Draw a **Hexagon** that has the same center as the circle/arc and is the approximate size shown in the figure. The *Hexagon* icon is usually stacked

under the *Rectangle* icon. (Your hexagon will contain many constraints that are not shown in the figure.)
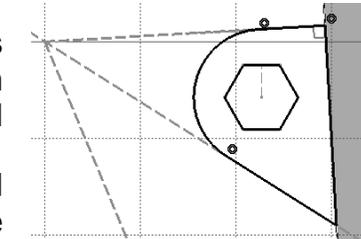
26) Deselect all.

27) Apply a dimensional **Constraint** to the distance between the flats of the hexagon as shown. To create this constraint, select the top line and then the bottom line. Double click on the dimension and change its value to *7 mm*.
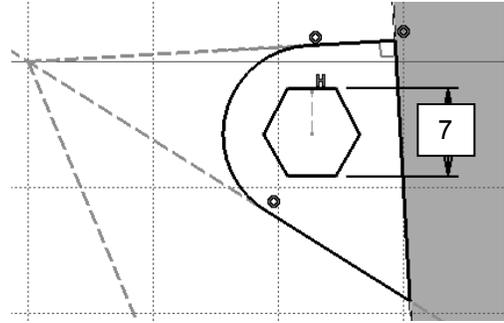
28) **Exit** the Sketcher and

Pad the sketch to a length of *2 mm* on each side (**Mirrored extent**).

Pad Definition

First Limit
Type: Dimension
Length: 2mm
Limit: No selection

Profile/Surface
Selection: Sketch.3
☐ Thick
Reverse Side
☑ Mirrored extent
Reverse Direction

More>>

OK   Cancel   Preview
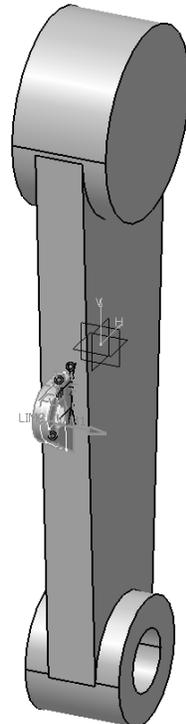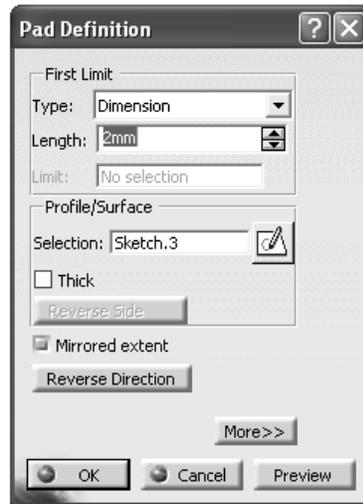
Section 4:   Creating geometrical constraints.

1) Enter the **Sketcher** on the flat face of the large cylinder.

Face/Pad.1/PartBody

Sketch face
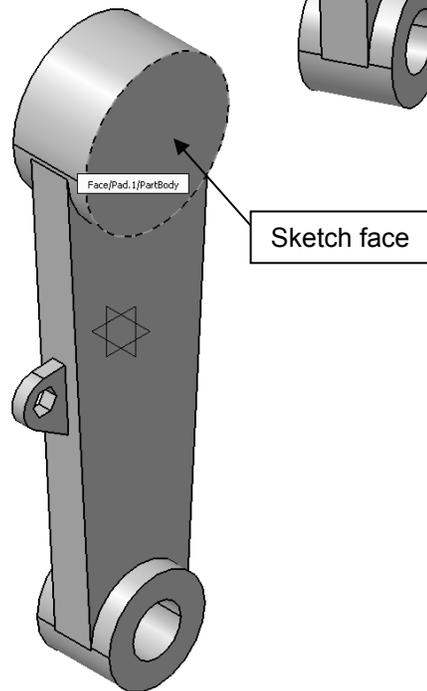
2) Deactivate the **Geometrical Constraint** icon (it should be blue). This will allow you to create profiles with no automatically applied constraints.

3) On the face of the large cylinder, draw the **Profile** shown. No geometrical constraints should be indicated.

Horizontal constraint

Vertical constraint

Parallel constraint

Perpendicular constraint
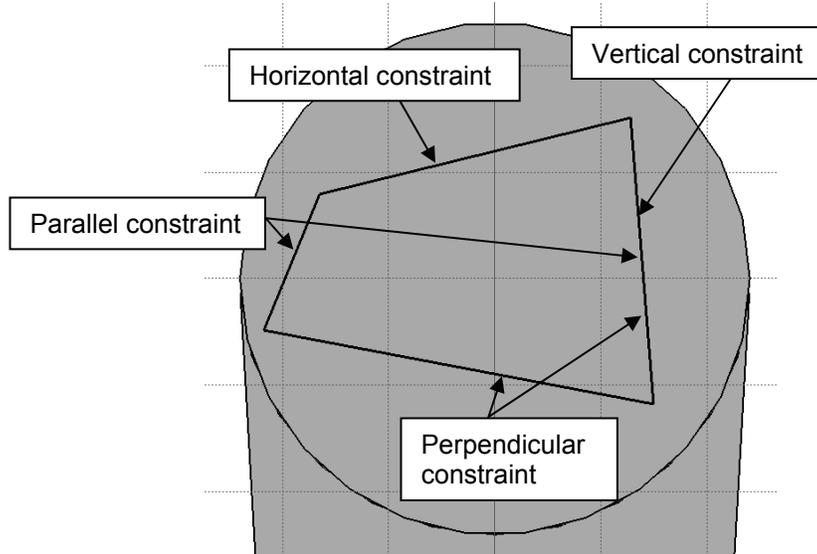
4) Deselect all.

5) Reactivate the **Geometrical Constraints** icon (it should be orange).

6) Apply a vertical constraint to the right line of the profile by right clicking on it and selecting **Line.? object – Vertical**.

7) Apply a horizontal constraint to the top line using a similar procedure.

8) Deselect all.

9) Apply a perpendicular constraint between the right and bottom line of the profile. Hold the CTRL key down and select the left and bottom lines. Select the **Constraints Defined in Dialog Box** icon. In the *Constraint Definition* window, check the box next to **Perpendicular** and then select **OK.**

10) Apply a parallel constraint between the left and right lines of the profile in a similar way.

Constraint Definition

- ☐ Distance
- ☐ Length
- ☐ Angle
- ☐ Radius / Diameter
- ☐ Semimajor axis
- ☐ Semiminor axis
- ☐ Symmetry
- ☐ Midpoint
- ☐ Equidistant point
- ☐ Fix
- ☐ Coincidence
- ☐ Concentricity
- ☐ Tangency
- ☐ Parallelism
- ☐ Perpendicular
- ☐ Horizontal
- ☐ Vertical

OK   Cancel
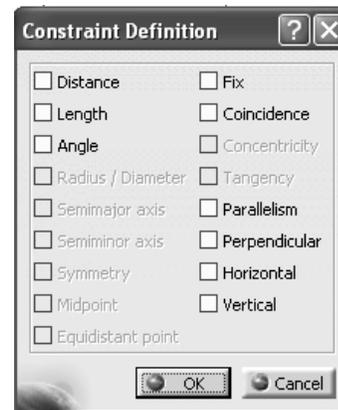
11) Apply **Constraints** [icon] to the rectangle and change their values to the values shown in the figure.
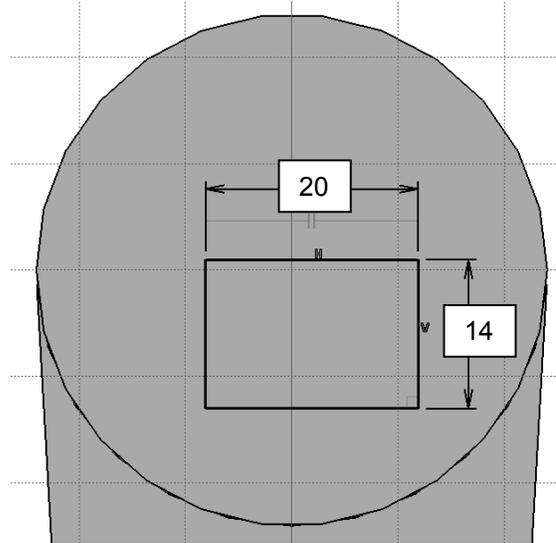
12) Apply the additional dimensional constraints shown in order to position the rectangle. Select the **Constraints** [icon] icon, then the circumference of the circle and then the appropriate side of the rectangle. Notice that once all the constraints are applied, the rectangle turns green indicating that it is fully constrained. If it did not turn green make sure the *Visualization of diagnosis* is activated in the *Options* window. (Tools – Options…)
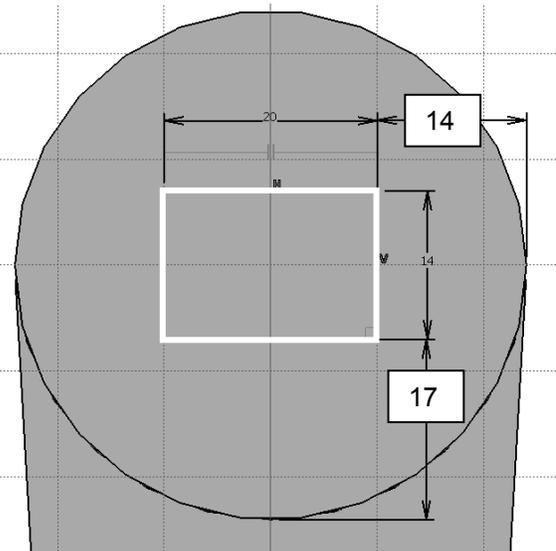
13) Draw the triangle shown using the **Profile** [icon] command. When drawing the triangle make sure that the top point is aligned with the origin ([icon]) and the bottom line is horizontal (H).
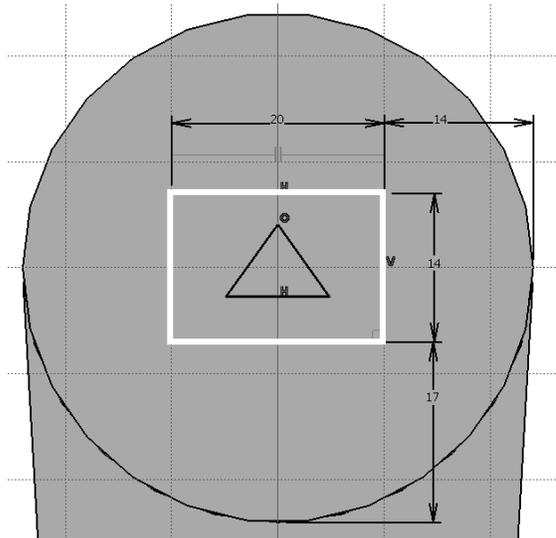
14) Constrain the vertical height of the triangle to be **6 mm**. Select the **Constraints** [icon] icon, select one of the angled lines of the triangle, right click and select **Vertical Measure Direction** and place the dimension.
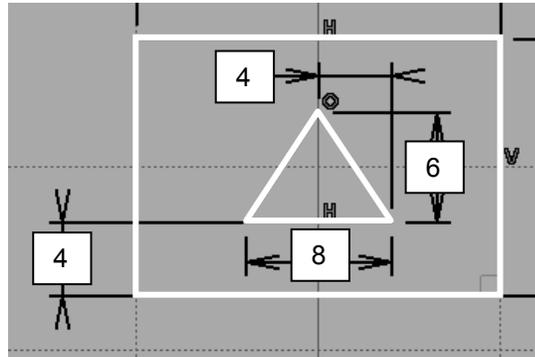
15) **Constrain** [icon] the rest of the triangle as shown.

16) **Exit** the Sketcher [icon] and **Pad** [icon] the sketch to a length of **5 mm.**

(Problem? If your sketch disappeared, *Copy* and *Paste* the sketch as described in the preface.)

17) **Save** your drawing.

Pad Definition

First Limit
Type: Dimension
Length: 5mm
Limit: No selection

Profile/Surface
Selection: Sketch.4
☐ Thick
Reverse Side

☐ Mirrored extent
Reverse Direction

More>>

OK   Cancel   Preview
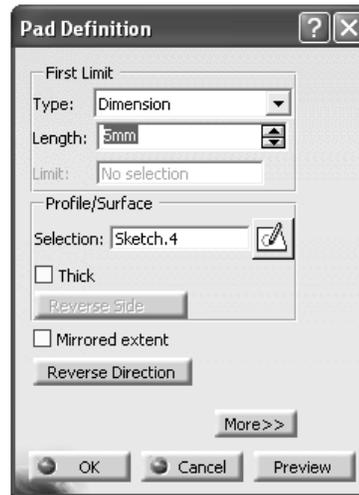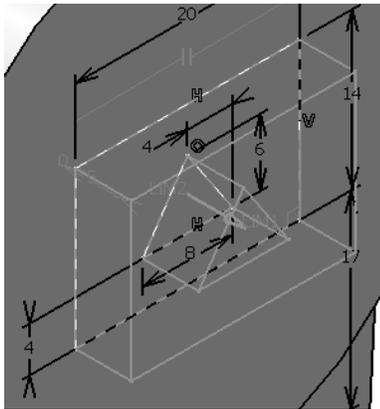
Section 5: Creating arcs.

1) Enter the **Sketcher** [icon] on the front face of the middle section.

Sketch face

Face/Pad.2/PartBody
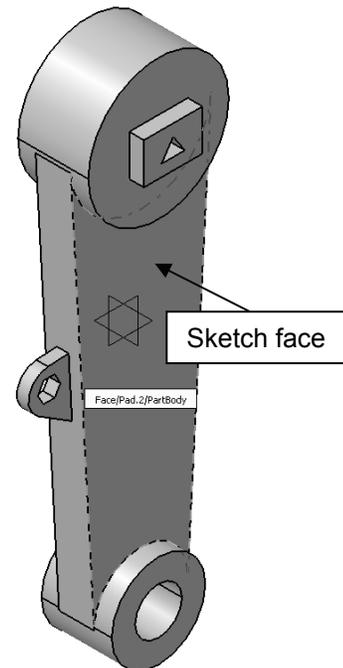
2) Activate the **Construction/Standard Element** icon.

3) Select the **Project 3D Elements** icon and then project the front face of the middle section.

4) Deselect all.

5) Deactivate the **Construction/Standard Element** icon.

6) Activate your **Snap to Point** icon.

7) Draw the profile shown. Use the **Three Point Arc** command to create the bottom arc, the **Arc** command to create the top arc. The *Arc* icons are stacked under the *Circle* icon. For assistance in creating the arcs, read the prompt line at the bottom of the graphics screen.

Use the **Profile** command to create the connecting lines.

Arc

Center point for arc

Three point arc

8) **Exit** the Sketcher and **Pad** the sketch to a length of *30 mm.*

Pad Definition

First Limit

Type: Dimension

Length: 30mm

Limit: No selection

Profile/Surface

Selection: Sketch.5

☐ Thick

Reverse Side

☐ Mirrored extent

Reverse Direction

More>>

OK   Cancel   Preview
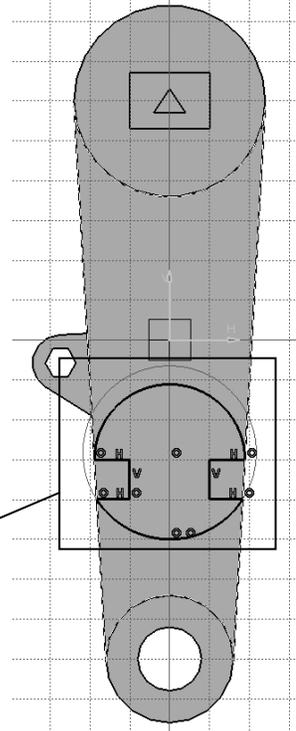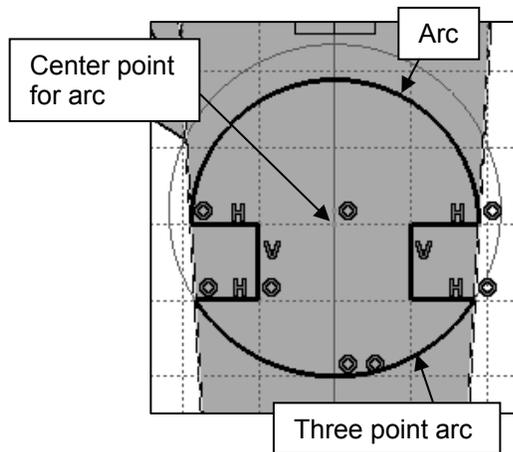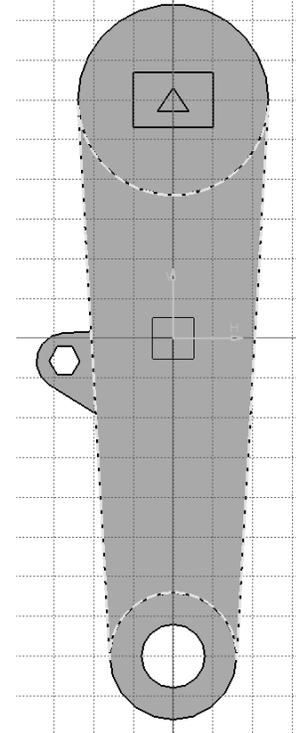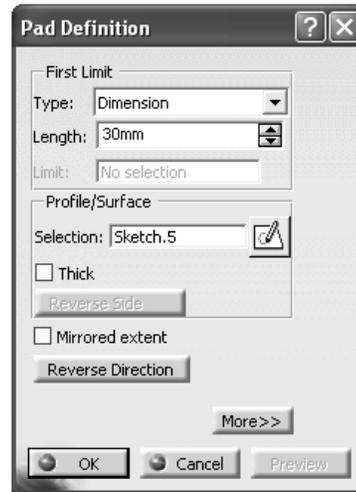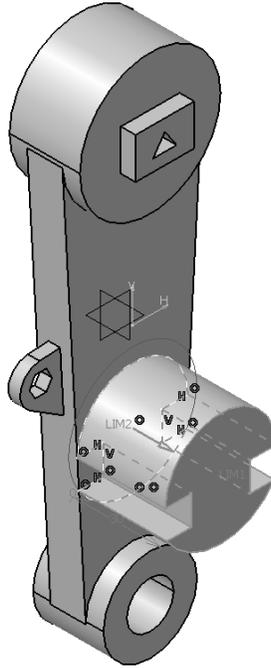
9) Deselect all.

10) Mirror the entire solid. Select the **Mirror** icon in the *Transformation Features* toolbar. Select the mirror element/face. In the *Mirror Definition* window select **OK**.

Mirror Definition

Mirroring element: Pad.5\Face.1

Object to mirror: Current Solid

OK   Cancel

Face/Pad.5/PartBody

Mirroring element