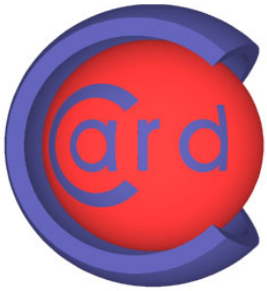

- **CCARD Ltd** is an independent consultancy offering CAE hardware and software installation and customisation services, specialising in CATIA and I-DEAS systems.

- CCARD also specialises in **Electronic Data Interchange** (EDI), and supplies, installs and supports OFTP/Odette based ISDN or TCP/IP solutions.

- In order to continue to provide its customers with the best products and support, CCARD has negotiated exclusive access to a **CATIA V5** Introduction User Guide, of unique quality and effectiveness, which is ideal as a **cost-effective self-study tutorial**.

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This extract from the CATIA V5 Introduction User Guide

- Includes the contents and index pages, together with the **full initial worked example**, overviews and summary of all examples, of the complete 134 page spirally bound manual.

- Provides an illustration of the style and content of this and other CATIA V5 User Guides compiled and published by **The CAD/CAM Partnership** - the leading independent CATIA specialist in the UK.

- Assumes the availability of a CATIA V5 workstation with a configuration license (such as 'MD2') and also familiarity with the CATIA V5 interface, such as use of the mouse buttons and command icons, in order to follow the initial worked example provided as an isolated sample.

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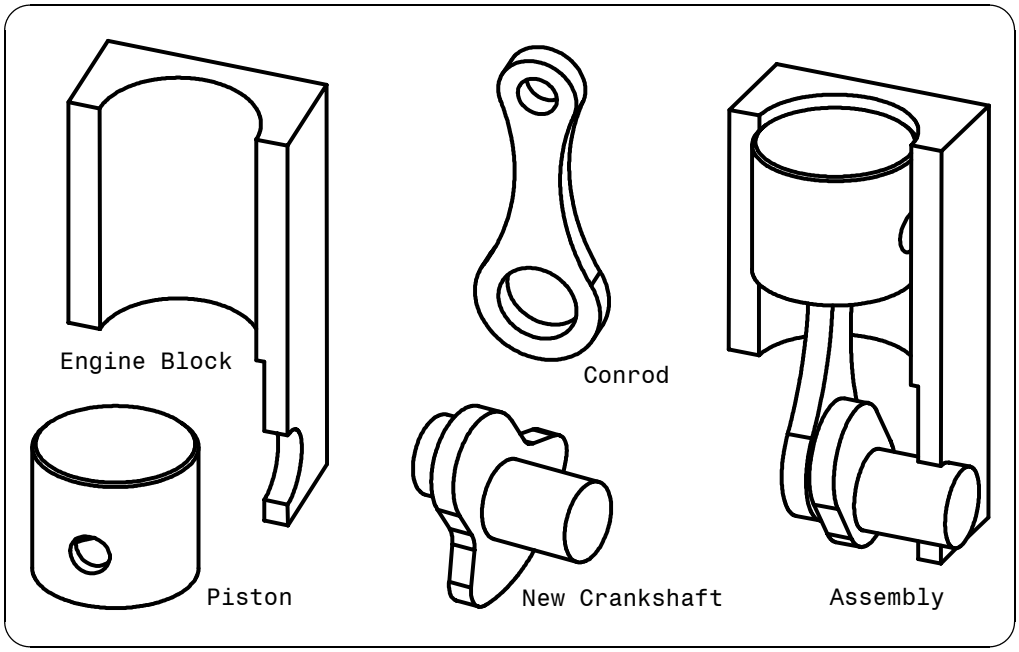
2. Overview Example

Engine Mechanism



Objective: To introduce the **Part Design**, **Assembly Design** and **Drafting** modules.

To model 3 simplified engine components, plus part of the engine block, so that the components can then be intelligently assembled, and animated. A drawing will be produced with views of the Crankshaft and the assembly, and the Crankshaft component subsequently modified to illustrate how the assembly model and the drawing reflect these changes.

Approach: A new **Part** document (file) will be created for each component.

The defining profiles will be created and dimensionally/geometrically constrained, typically in the yz 2D plane.

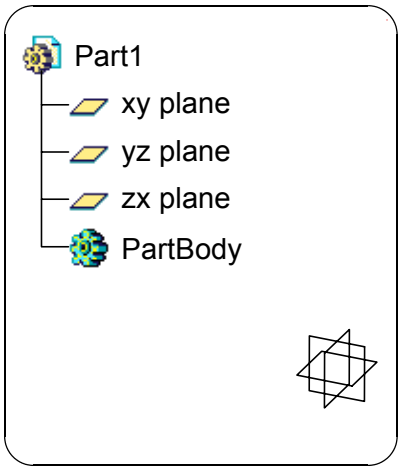
The assembly is a **Product** document which will reference the Part documents.

Comments: In this example, each component is created **independently**, i.e. no reference is made to geometry in the other Part documents. Changes to one Part document will therefore not affect the other Parts. (It is possible for changes in one Part to automatically be reflected in all related Parts).

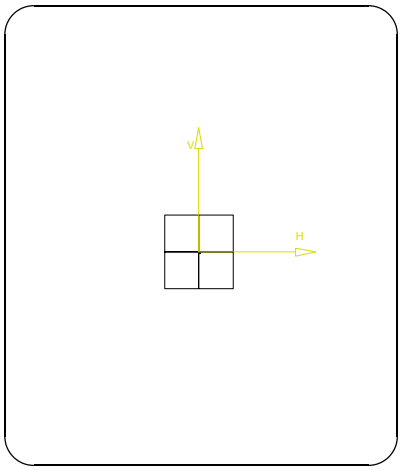
Overview Example

1. Conrod (Part)

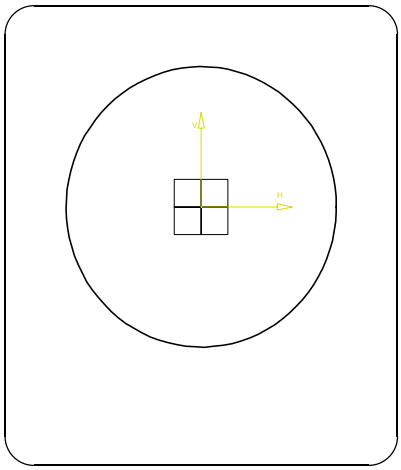
1. Activate Part Design



2. Choose working 2D plane



3. Create top hole



1. **Start** CATIA

   If the **Welcome to CATIA V5** window is displayed, then...
   Use **MB1** to activate ■ **Do not show this dialog at start up**
   Select **Close** to remove this window

   Close the proposed Assembly Design workbench window...
   Select **File** + **Close** to close the **Product1** window

   Select **Start** + **Part Design** to activate the Part Design workbench

   *A '**specification tree**' displays:* ***Part1*** *- the default new Part document name*
   3 datum planes *- which are also displayed as geometry*
   *An empty '**Body**' in which Part geometry can be created*

2. Select **Sketcher:** and then the **yz** reference **plane** (or vice-versa)

   Note: *The **yz** reference **plane** can be selected as geometry, or via the specification tree*

   The Sketch plane, by default, rotates to display normal to the screen, as shown

3. **Double-click** (so as to use more than once) **Circle:**

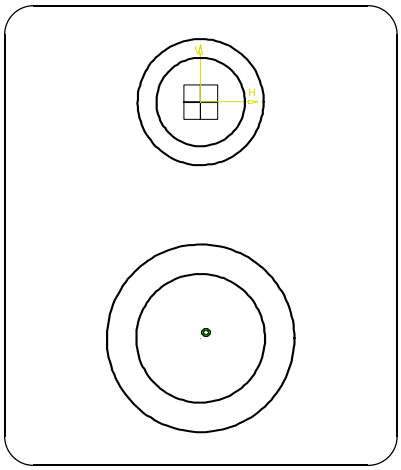
   First create the top hole for the gudgeon pin...

   Select the **origin point** (a temporary blue circular icon must be displayed) as the **centre**

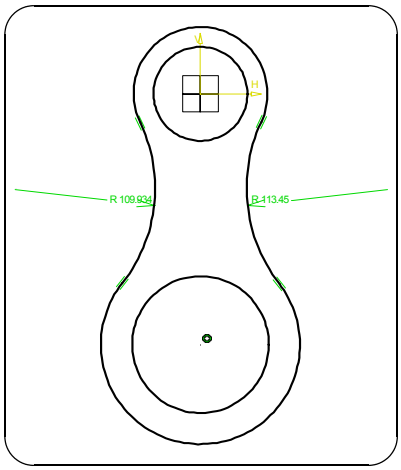
   Indicate (using MB1) **any point** at the approximate location of the circle circumference

1. Conrod (Part) *(continued)*

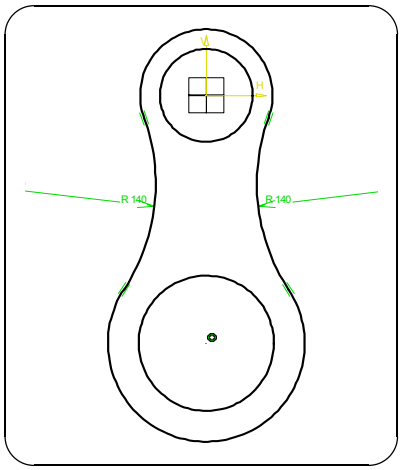
4. Create other circles



5. Fillet left then right



6. Correct the radius values



4. Similarly, create a concentric circle by first selecting the **existing centre point**

(**Hold MB2** and **click MB1**, and then **move** the mouse **vertically** to zoom out as required)

Create a third circle by first selecting a centre point **vertically below** the origin
(a temporary blue vertical line must be displayed below the V axis)

Note: *The '**Coincidence**' of this lower centre point with an extension of the vertical axis is automatically created as a geometrical constraint (the green circle symbol), but **only if** the **Geometrical Constraints** icon:*  *is **active** (the default setting)*

Create another concentric circle by first selecting the **second centre point**

5. **Double-click Corner:** 

Select each outer circle and indicate a point to approximately define the **left** fillet curve

Similarly define the **right-hand** fillet curve

Note: *Should the radii Constraint values be displayed with a ± tolerance symbol, then select the **Tools** + **Options**... + **Parameters and Measure Parameters Tolerance** tab, and deactivate ☐ **Default tolerance** (for future Constraints)*

6. **Double-click** a fillet curve dimension value
Enter the required value (**140**mm) in the Constraint Definition window

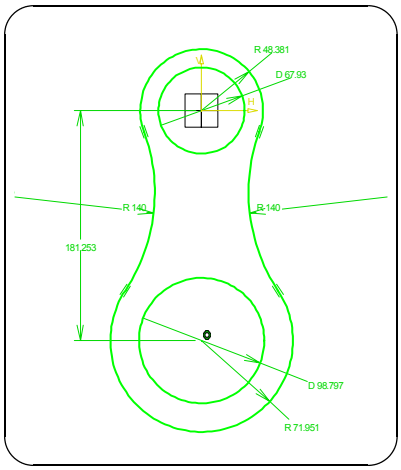
Similarly correct the other fillet curve dimension value

NOTE: *As geometry is fully constrained it turns from white to green. White geometry therefore indicates that additional geometrical or dimensional constraints are required to completely specify the profile...*

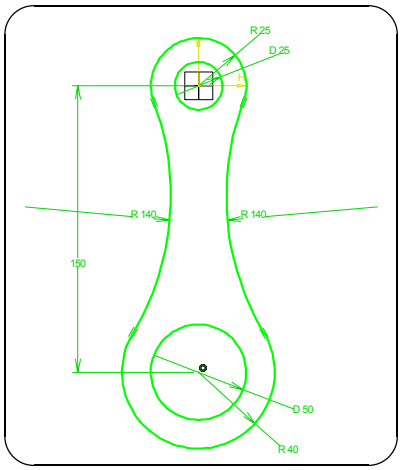
Overview Example

1. *Conrod (Part)* *(continued)*

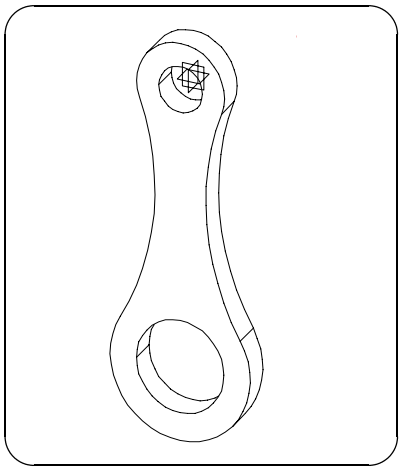
7. Constrain the profile



8. Correct the dimensions



9. Create the solid Pad



7. **Double-click Constraint:** 

   Select each **circular curve** and indicate to create radius and hole diameter dimensions
   Note: *These dimensional constraint values will be arbitrary, for example as shown*

   Select the **horizontal axis** and the lower centre **point** to create the vertical dimension

8. **Double-click** each dimension value, and enter the required value (as shown)
   (Top: radius **25**mm and Ø**25**mm, vertical offset **150**mm, lower: radius **40**mm and Ø**50**mm)

   Select **Exit:**  **to leave the Sketcher and to enable 3D geometry creation...**

9. Select **Pad:**  and in the Pad Definition window...
   Specify a Length of **16**mm (in the positive X direction)

   To define the material of the part...
   Select the **Part** and then select **Apply Material:**  (or vice versa)
   Select **Metal + Steel** from the material Library window and select **OK**

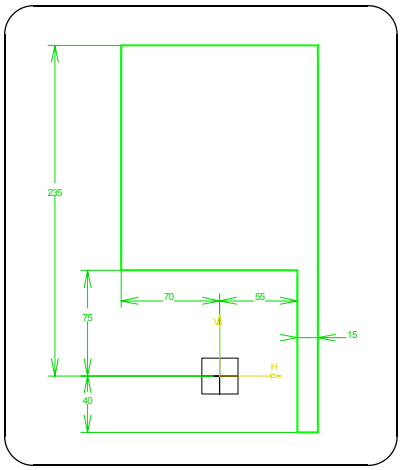
   Use **MB3** to select  **Part1** from the specification tree
   Select  **Properties**
   Select the **Mass** tab to review the **Mass** Properties
   Select the **Product** tab, and change the **Part Number** to ***Conrod***
   Select **OK**

10. Select **File + Save As...** to display the **Save As** window
    Save the Part as ***Conrod*** in the default location (e.g. *E:\Catdata\My_work)*
    Select **File + Close** to close the Conrod document

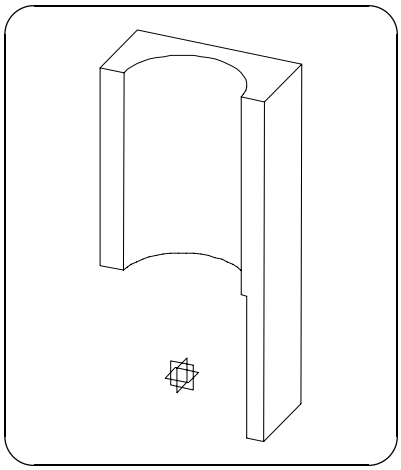
Chapter 2

2. Block (Part)

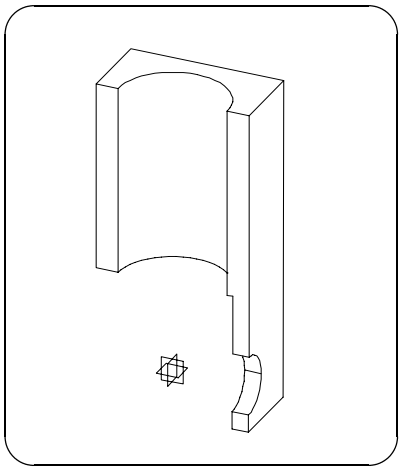
1. Create block profile



2. Create cylinder shaft



3. Create crankshaft hole



Part of an engine block will be required to support the intelligent assembly of the engine parts...

1. Select **Start** + **Sketcher** and select the **yz** reference **plane**
   Note: *Starting the Sketcher workbench creates a new Part document*
   Select **Profile:** and create orthogonal line segments, ending at the first point
   Double-click Constraint: to create dimensional constraints for the profile
   Double-click each dimension value, and enter the correct/required value (as shown)

2. Select **Exit:** and then select **Pad:**
   Create a prism with a (**-X** direction) depth of **70**mm

   For a new Sketch (you can select the command icon and then a plane, or vice-versa)...
   Select **Sketcher:** and then the **xy** plane (or an existing face parallel to the xy plane)
   Select **Circle:** to define a Ø**100**mm circle centred at the origin (and then **Exit:**)
   Select **Pocket:** with Limit **Type** set as **Up to last** (**Reverse Direction** if required)

3. Create a new Sketch containing a Ø**50**mm **circle** centred at the origin on the **zx** plane
   Use **Pocket:** with Limit **Type** set as **Up to last** (**Reverse Direction** if required)

4. ***Optionally****, to define the material of the part...*
   Select the Part and then select **Apply Material:** (or vice-versa)
   Select **Metal** + **Aluminium** from the Library window

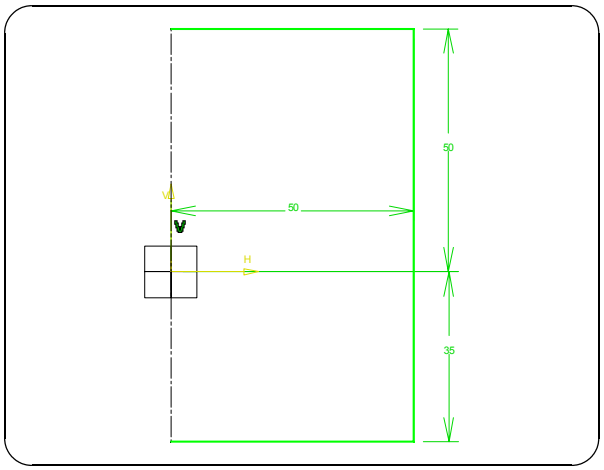
   Use **MB3** to select **Part1** via the specification tree, and select **Properties**
   Change the Product **Part Number** to ***Block***
   Select **OK**

   Select **File** + **Save As...** to save the Part as ***Block***
   Select **File** + **Close** to close the Block document

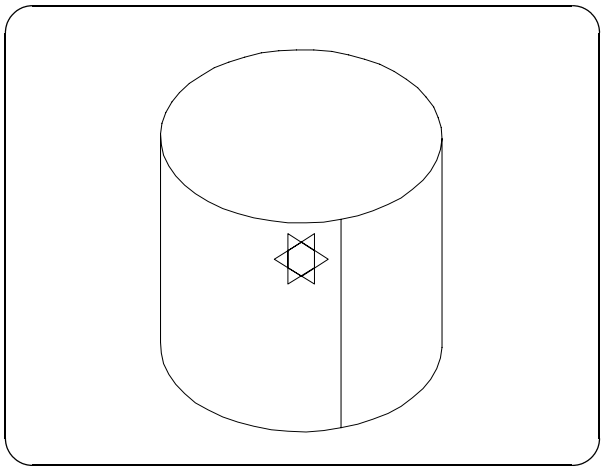
Overview Example

3. Piston (Part)

1. dimension cylinder profile with axis



2. Create 360° solid of revolution



1. Select **New:** and **Part** and then select **OK**
   (This is yet another way of starting a new Part)

   Select the **yz** reference **plane** and then **Sketcher:** (or vice-versa)

   Select **Profile:** and create line segments with endpoints inline with the vertical axis

   Note: *To end the definition of a Profile (which does not finish at its start point), reselect (to deactivate) the* ***Profile:*** *command icon.*

   Select **Axis:** and define a line joining the endpoints
   (Press **MB1 to deselect** the line)

   Double-click Constraint: and create the **3** distance dimensions **from the axis**
   Double-click each dimension value, and enter the required value (as shown)
   (Vertical offsets **50**mm and **35**mm, and horizontal offset **50**mm)

   Note: *The Axis line is* ***not*** *fixed (it could be dragged away from the vertical axis)*

2. *To create a solid of revolution from the Sketch profile...*

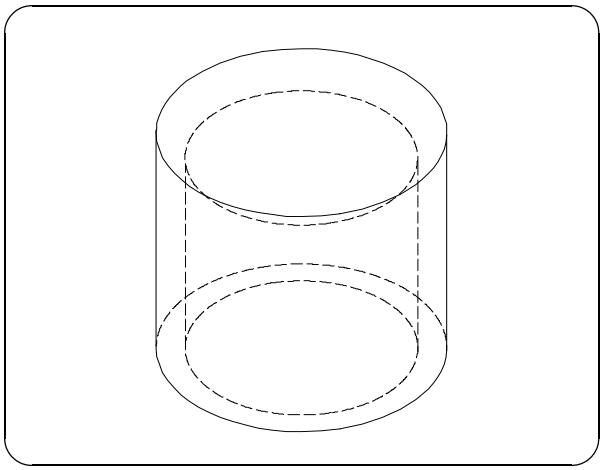
   Select (**Exit:** and) **Shaft:**

   Note: *If the Sketch did not contain an Axis type line, then it would be necessary to select an axis of revolution, for example the vertical (V) axis.*

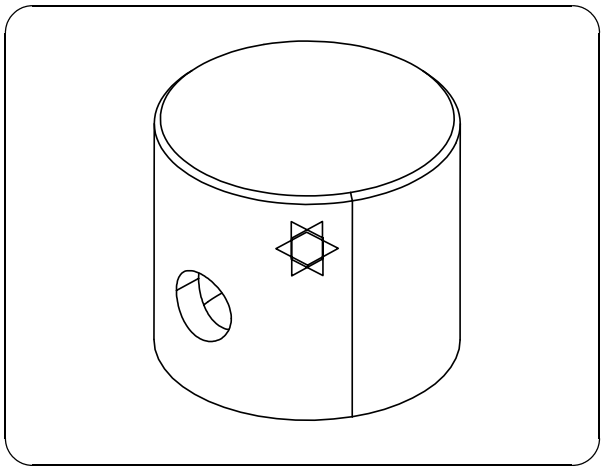
   Verify that the proposed **First angle** limit is **360°**, and select **OK**

3. Piston (Part) *(continued)*

3. Define shell thickness operation



4. Create gudgeon pin hole and 2mm chamfer



3. Select **Shell:**
   Select the bottom **face** (for removal)
   Enter an **Inside thickness** of **10**mm

4. Select the **yz** reference **plane** and then **Sketcher:** (or vice-versa)
   Select **Circle Using Coordinates:**
   Define a **12.5**mm radius circle at coordinates **0,0**

   Note: *This circle centre point is fixed - but independent of the origin point. The location of the circle centre can be moved by modifying the offset dimensions.*

   Although the circle has been created efficiently, the circle is unlikely to be moved from the origin in this example, and therefore would have been more appropriately created as a ***Circle:*** *using the origin as its centre point.*

   Select **Exit:** and then select **Pocket:**
   Select **More>>** to set both Limit **Types** to **Up to last**
   Select **OK**

   Select **Chamfer:** and the top face or edge (or vice versa)
   Define a **2**mm chamfer at **45°**

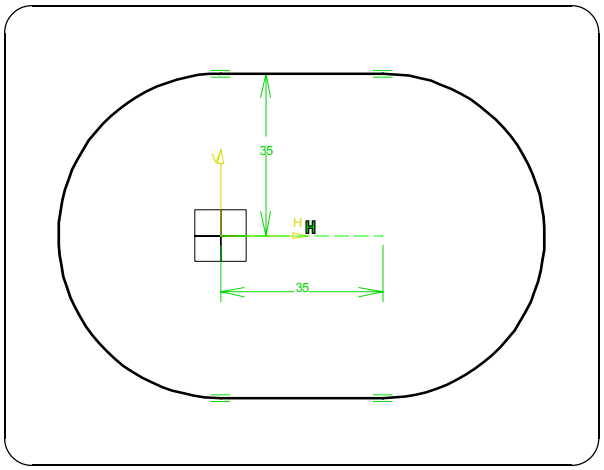
5. Use **MB3** and **Properties** to change the Product **Part Number** to ***Piston***

   Select **File** + **Save As**... to save the Part as ***Piston***
   Select **File** + **Close** to close the Piston document

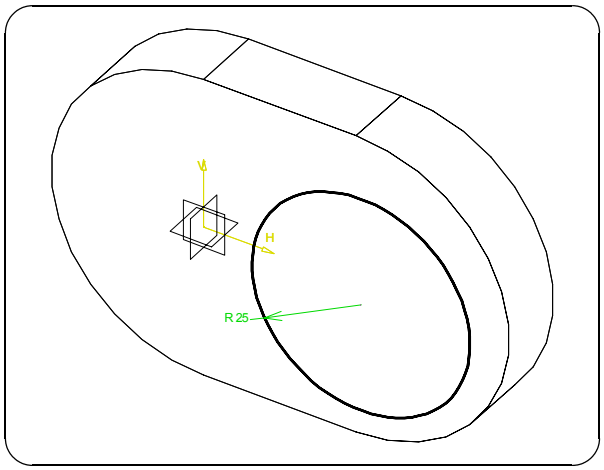
Overview Example

4. Crankshaft (Part)

1. Define the 35mm oblong profile



2. Create Pad and define an R25mm circle



1. Select **Start** + **Sketcher** and then select the **yz** reference **plane**
(This is the most efficient way to start a Sketch for a new Part)

NOTE: *Rather than literally sketch geometry, and then return to constrain and correct the arbitrary dimension values, it is possible to* ***enter specific values*** *via the* ***Sketch tools*** *toolbar menu (which is the one including the Grid icon).*
(To ensure that all of the numerical value entry fields of this menu are visible, then it should be dragged into the main window to create a separate window)

Change the 'type of profile' (i.e. from or etc.) to **Oblong:**

First select the origin **point**
Enter a length **L:** of **35**mm and then select a **point** horizontally to the right
Enter a radius **R:** of **35**mm

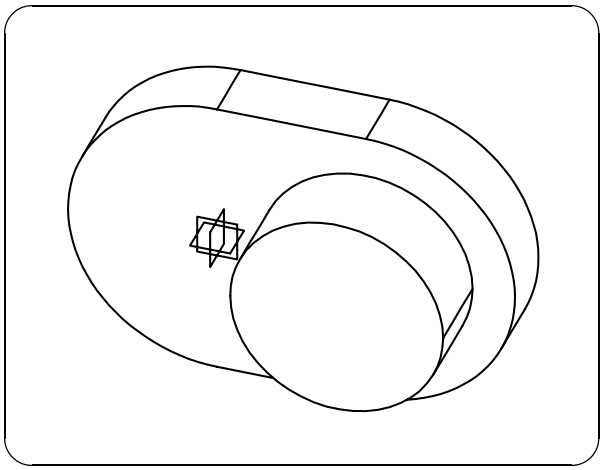
2. Select **Exit:** and then **Pad:**
Enter a **Length** (i.e. a 'thickness', in the **-X** direction) of **16**mm

Select **Sketcher:** and then the front **face** of the pad (or vice-versa)

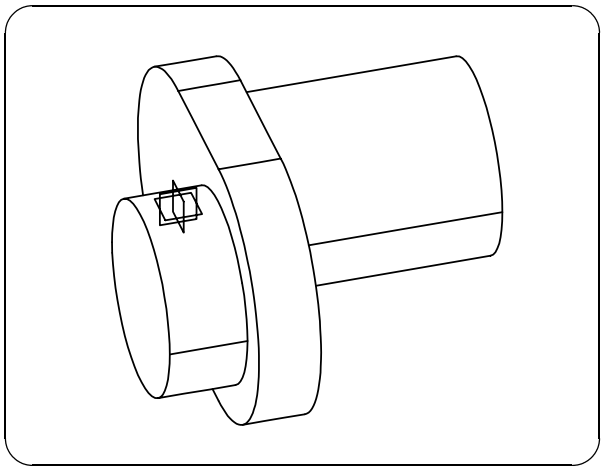
Select **Circle:**
Select a centre **point** (anywhere!) and enter a radius **R:** of **25**mm

4. Crankshaft (Part) *(continued)*

4. Create concentric R25 x 20mm cylinder



5. Create an R25 x 60mm cylinder



4. *The circular profile can be defined to be always concentric with an edge of the Pad...*

With the circle still highlighted as the current element...
Select **Constraint:**
Select the semi-circular right-hand **edge** of the Pad
Select the proposed dimension **value**, using **MB3**, to replace it with a **Concentricity**

Select **Exit:**, select **Pad:**, and then enter a **Length** (thickness) of **20**mm

5. ***NOTE:*** *Whenever a working **plane** for a **Sketch** is defined, it may be **rotated** to be parallel to the screen so as to be viewed orthogonally. On leaving the Sketch, the previous (typically isometric) viewpoint will be reinstated. This automatic presentation of an orthogonal viewpoint does not help to clarify the location of the Sketch, and is therefore optional...*

Select **Tools + Options... + Mechanical Design + Sketcher**, and in the **Sketcher** tab...
Deactivate Sketch Plane ☐ **Position sketch plane parallel to screen**
Select **OK** to close the Options window
(Normal View:** and **Isometric View:** can obtain the same effect **when required)

Select **Sketcher:** and then the **rear face** of the 16mm pad (or vice-versa)
Create a **25**mm radius **circle** centred at the origin

Select **Exit:**, select **Pad:**, and then enter a **Length** (thickness) of **60**mm

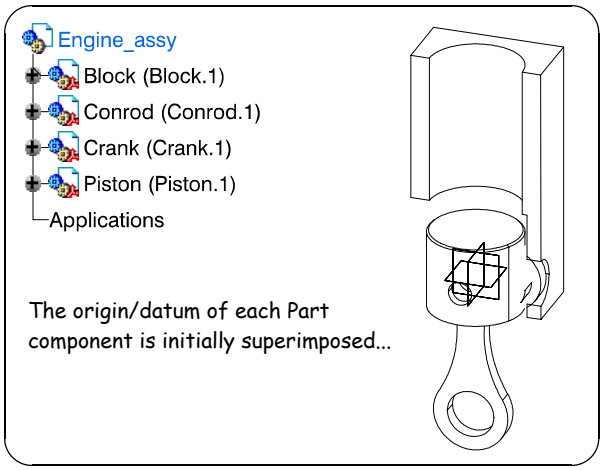
6. Use **MB3** and **Properties** to change the Product **Part Number** to ***Crank***

Select **File + Save As...** to save the Part as ***Crankshaft***
Select **File + Close** to close the Crankshaft document

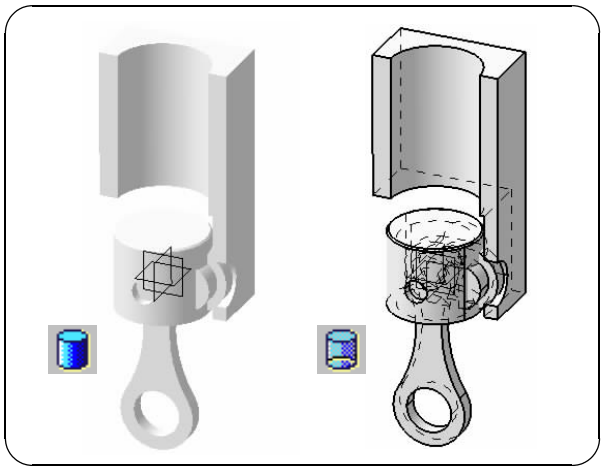
Overview Example

5. Engine Assembly (Product)

1. Assemble existing components



2. The display mode can be changed



1. Select **Start** + **Assembly Design** to create a new Product (assembly) Document

   Select ***Product1*** using **MB3** and select **Properties**
   Select the **Product** tab and change the **Part Number** to ***Engine_assy***

   To assemble the required components...
   Select **Existing Component:** (to be inserted into the current Engine_assy Product...)

   Note: *Alternatively,* ***Existing Component with Positioning:*** *will additionally both place and constrain a component. However, the more flexible approach is to create the relationships between components later, particularly in this example, where the position/orientation of the components are interrelated...*

   Ctrl-select the **Piston, Crankshaft, Conrod** and **Block** Part documents
   Select **Open**

2. *Note that at any time the Display Mode can be changed...*
   Select the current setting, for example, **Shading with Edges:** and...
   Select **Shading (SHD):** for shading without edges
   Similarly change the Display Mode to **Shading with Edges and Hidden edges:**

   Note: *To obtain the Hidden Line Removal type display used to illustrate this manual:*
   Select **Customize View Parameters:** for the Custom View Modes window
   Activate ■ **Dynamic hidden line removal** and then select **OK**

   Select **Shading with Edges:**, or, **Shading with Edges without Smooth Edges:**

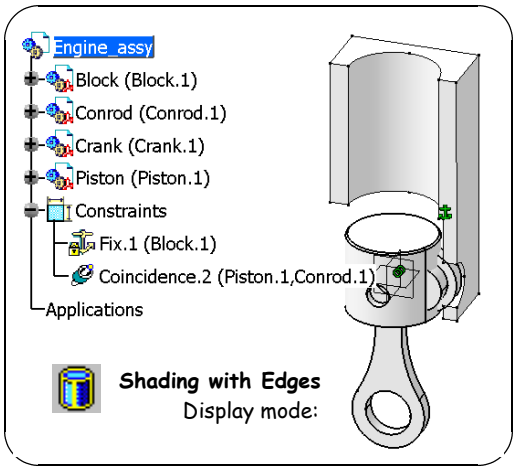
   Note: ***Customize View Parameters:*** *also provides an option for another similar display, but with* ● ***Half visible smooth edges****, for a less prominent display of internal 'edges' at tangential boundaries*

Caution: *Engine_assy must remain as the 'current object' in the Specification Tree, otherwise, if instead a Part is current, then the Constraints Toolbar will be dimmed/unavailable...*

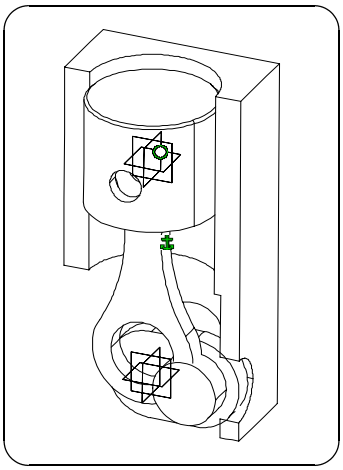
Chapter 2

5. Engine Assembly (Product) *(continued)*

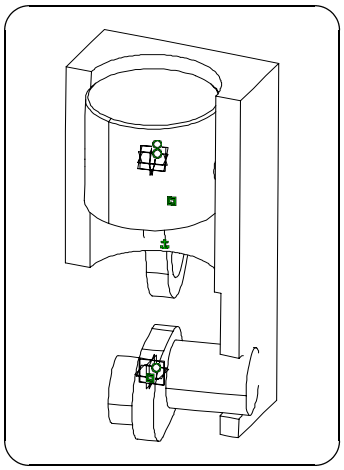
3. Fix Block and link Piston/Conrod



4. Move Piston/Conrod



5. Link Parts and rotate



3. Select **Fix Component:** and then the ***Block*** Part (or vice-versa)

   Note: *By default the component will be fixed* ***absolutely*** *- as indicated by a lock symbol on the* ***fix*** *icon in the specification tree.*
   *(**MB3 + Properties** and the **Constraint** tab and deactivating ☐ **Fix in space** would instead define the Fix to be only relative to other referenced components)*

   Select **Coincidence Constraint:** (the geometry may have to be rotated first…)
   Select the ***Piston*** hole (horizontal proposed ***axis***) and the ***Conrod*** top hole (proposed ***axis***)

   NOTE: *The Piston (**first** selected Part) would have **moved to** share axis with the Conrod (**second** selected Part) **if required** unless the first Part had been previously fixed*

4. Select **Manipulation:** (**or**, use the **Compass** and **Shift** to drag both components)
   Select **Drag along Z axis** and set ■ **With respect to constraints**
   Select either the Conrod or Piston Part, and drag the pair vertically upwards

5. Select **Coincidence Constraint:**
   Select the ***Piston*** (proposed axis) and then the ***Block*** shaft (proposed axis)

   Note: *The Crankshaft, Conrod and Piston must be rotated together through 90° clockwise before the Crankshaft can be linked to the Block…*

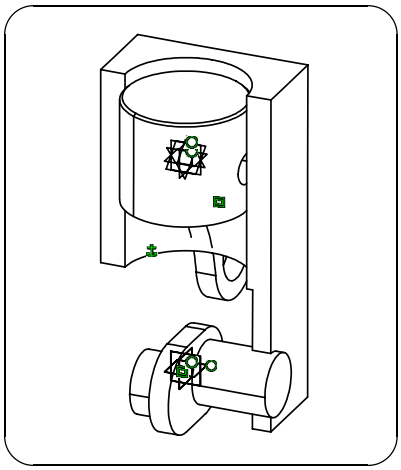
   Select **Contact Constraint:**
   Select the ***Conrod*** (**rear** face) and then the ***Crankshaft*** (**front** face), or vice-versa

   Select **Manipulation:**
   Select **Drag around Z axis** and set ■ **With respect to constraints**
   Select the ***Crankshaft*** and drag so as to rotate the 60mm cylinder towards the hole
   Note *that this is* ***very approximate*** *so that part of the hole remains for selection!*

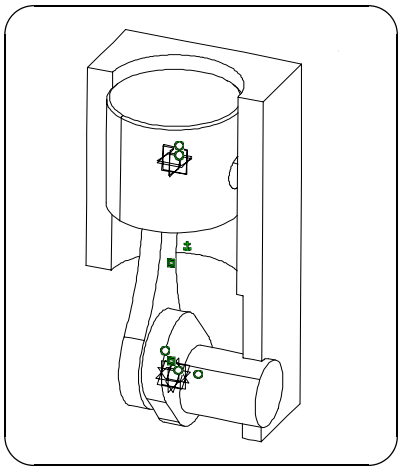
Overview Example

5. Engine Assembly (Product) (continued)

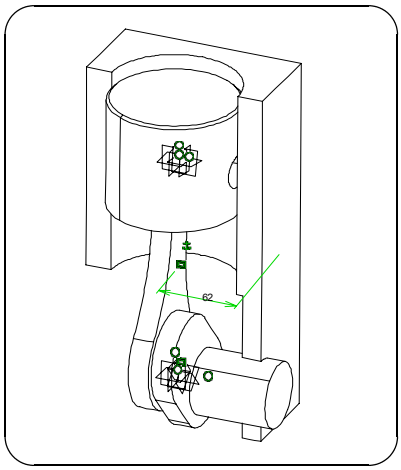
6. Define Crank rotation



7. Link Conrod/Crankshaft



8. Centre the Conrod



6. **Double-click Coincidence Constraint:**

   To align the Crankshaft with the Block...
   Select the ***Crankshaft*** (60mm Cylinder) and then the ***Block*** (horizontal hole)

7. *To connect the Conrod and Crankshaft...*
   Select the ***Crankshaft*** (20mm Cylinder) and then the ***Conrod*** (lower hole)

8. *The Conrod/Crankshaft are not correctly centred with respect to the Piston...*
   Select **Offset Constraint:**
   Select the ***Conrod*** (face) and then the ***Block*** (outer face)
   Enter an **Offset** of **62**mm

9. Select **File + Save As...** (or **Save**), or select the equivalent **Save:** command icon

   If a **Save Management** window is presented, then select **Save As...** for the ***Engine_assy***

   *A **Save As** window is always presented the first time a document is saved...*
   Select **OK** to confirm that the Product document is to be saved as ***Engine_assy***

   Optionally temporarily change the Display Mode to **Shading with Material:**

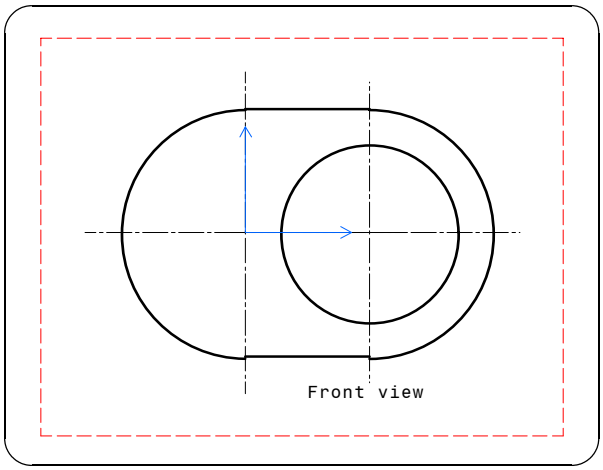
   Select **Manipulation:**
   Select **Drag around any axis** and activate ■ **With respect to constraints**
   Select (the proposed **axis** of) the Crankshaft (60mm Cylinder)
   Select the Crankshaft and drag so as animate the piston mechanism

   Select **File + Close** to close (without saving) the **Engine_assy** window

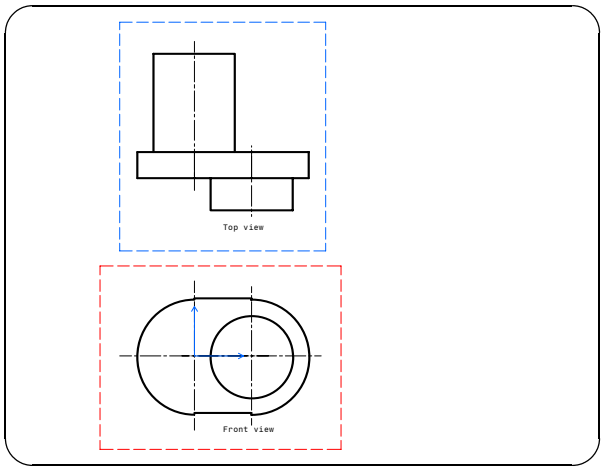
Chapter 2

6. Drawing Generation

1. New sheet with front view



2. Define plan view



1. **Open:** the **Crankshaft** Part

   Select **Start** + **Drafting** (+ **Empty sheet**) and **Modify...**
   Verify that the **Standard** is **ISO** and set the **Format** to **A2 ISO** *(594x420mm)*
   Also verify that **Orientation** is **Landscape** and **Scale of sheets** is **1**
   (Select **OK** in the New Drawing window and **OK** in the Create New Drawing window)

   To determine the Projection Method, select **Sheet.1** using **MB3** + **Properties**
   Verify that the ● **Create projection views using third angle standard** option is current
   Select **OK**

   Deactivate the **Sketcher grid:** and **Snap to point:** options

   Select **Window** + **Tile Horizontally** to display both Crankshaft.CATPart and Drawing1 windows - *(the currently active window always displays above the other window)*

   Select **Tools** + **Options...** + **Mechanical Design** + **Drafting** and the **Layout** tab
   Deactivate **View Creation** ☐ **Scaling factor** (the display of the View Scale)
   Select the **View** tab
   Verify that the ■ **Generate axis** and ■ **Generate center lines** options are active
   Select **OK**

   Select **Front View:** and then the **front face** of the ***Crankshaft***
   Optionally select the view frame (using MB1) and drag to a more appropriate location
   Select the proposed **view** geometry to generate the **Front** view

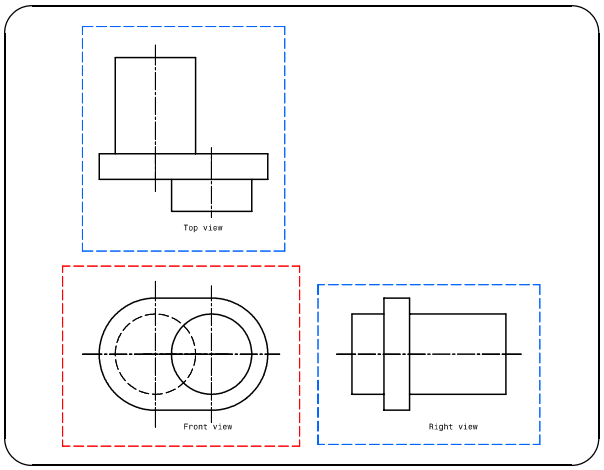
2. Select **Projection View:** *(available in same the group of icons as Front View)*
   Position the cursor **above** the Front View
   Select the proposed view to generate the **Top** view

   Note *that the* ***Front*** *view is still the* ***current*** *view (with the red frame)...*

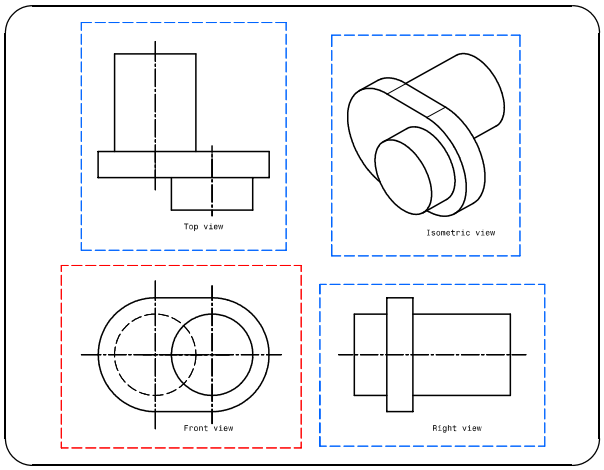
Overview Example

6. *Drawing Generation* *(continued)*

3. Define side view



4. Define isometric view



3. Select **Projection View:**
Position the cursor **to the right** of the Front View
Select the proposed view to generate the **Right** view

4. ***Note:*** *An isometric view is created with the same orientation as the selected Part, i.e. a* ***standard isometric is not necessarily created****. If a standard isometric view is required, then the Part orientation must first be defined by selecting* ***Isometric View:*** *(from within the* ***Part*** *window/workbench)*

Select **Isometric View:** (from within the **Drawing** window/workbench)
Select any face of the ***Crankshaft Part***
Optionally modify the default isometric orientation via the **compass**
Select the proposed view (**or** the centre of the compass) to generate the **Isometric** view

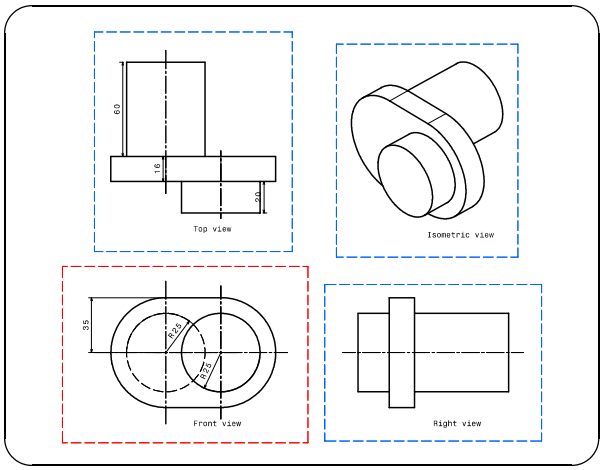
Select the **Drawing window** maximise icon: to display only the Drawing window
Select and drag the **Isometric** view **frame** to relocate the Isometric view as required

Select (using **MB3**) the **frame** of the **Front** View and select **Properties**
Within the **View** tab, activate the display of **Dressup ■ Hidden Lines**
Select **OK**
(The circle representing the rear cylinder is displayed dashed)

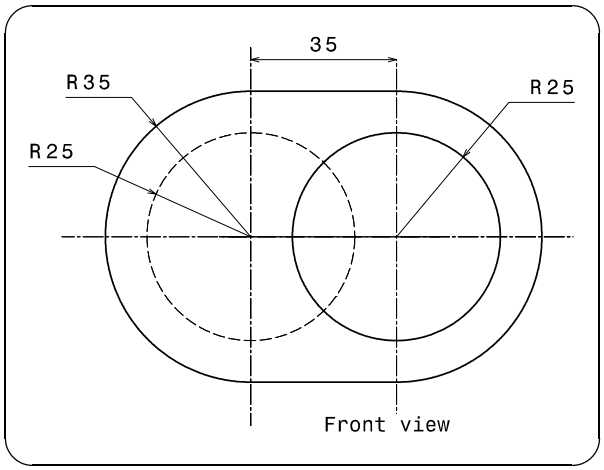
Select **Tools + Options... + Mechanical Design + Drafting** and the **General** tab
Deactivate **View axis ☐ Display in the current view**
(... which may only become effective when you next change the current view)

6. Drawing Generation *(continued)*

5. Automatically generated dimensions



6. Redefine the dimensions in the front view



5. Select **Generating Dimensions:**

*The **Generated Dimensions Analysis** window displays the number of constraints found (e.g. 7), and how many dimensions were created (e.g. 6) from these constraints.*
Select **OK**

6. *In the **Top** view:*
Select the **60**mm dimension (**value**) and **then** drag the **dimension line** to the left
Ctrl-select the **16** and **20**mm **dimensions**
Use **MB3** to select **Line-Up** and select the **60**mm **dimension** as the reference
Verify that Offset to reference is **0** (in the **Line Up** window) and select **OK**

*In the **Front** view:*
Select and **Delete** the vertical **35**mm dimension

To be able to indicate the initial location of a dimension as it is created...
Select **Tools** + **Options...** + **Mechanical Design** + **Drafting** and the **Dimension** tab
Activate the ■ **Dimension following the mouse (ctrl toggles)** and select **OK**

***Note** that small symbols will indicate the geometry (line, arc or circle) being detected...*
Select **Dimension:** and select the left-hand semicircle
Indicate a point to locate the radius dimension
Change the **Dimension Line** format from to
(via the Dimension Properties toolbar - displayed at the top of the screen)

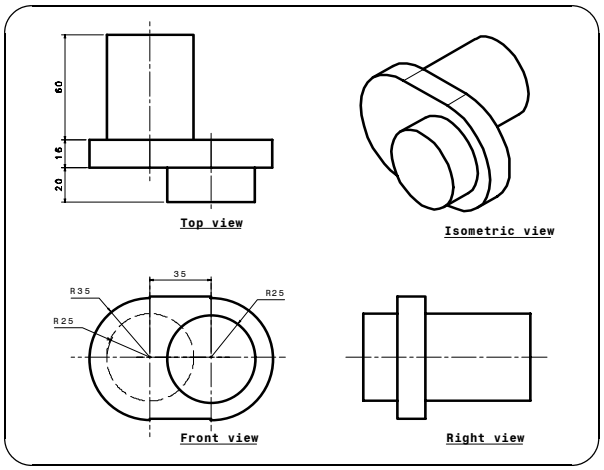
Select **Dimension:** to create the 35mm dimension between the 2 vertical centrelines

Select and relocate both of the 25mm radius dimensions and their values, and...
Change their **Dimension Line** format to

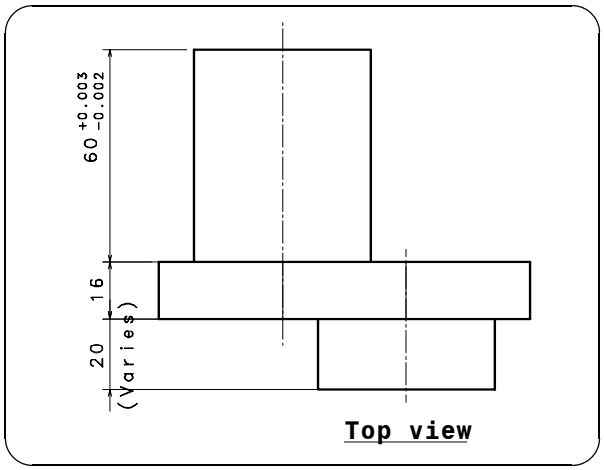
Overview Example

6. *Drawing Generation* *(continued)*

7. Modify view texts, and switch off frames



8. Define tolerance and associated text



7. **Ctrl-select** the 4 View title **texts**
   Set the **Font Size** to **5 mm** and select **Bold:** B and **Underline:** S

   To switch off the boundary frames...
   Ctrl-select the 4 View **frames** (or the 4 Views listed under Sheet.1)
   Use **MB3** to select **Properties**
   Within the **View** tab, deactivate **Visualisation and Behavior** ☐ **Display View Frame**

   Note: *In fact, the View Frames do not plot, and can optionally remain displayed - since they are the most efficient means of repositioning a View.*

8. *In the **Top** view:*

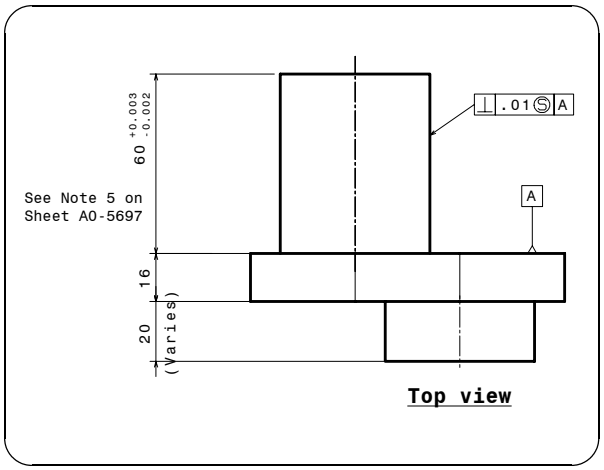
   Select (using **MB3**) the **60**mm dimension, and select **Properties**
   Within the **Value** tab...
   Set the **Format Precision** (Main value) to **0.001**
   Select the **Tolerance** tab and...
   Set the **Main Value** to the **TOL_NUM2** format
   Set the tolerances **Upper value: 0.003**, **Lower value: -0.002**,
   Select **OK**

   *To enable movement of the dimension value **only** along the dimension line...*
   Select **Tools** + **Options...** + **Mechanical Design** + **Drafting** and the **Manipulators** tab
   Activate **Move value:** during ■ Modification and select **OK**
   Select the dimension and then the **arrows symbol** to move the value vertically upwards

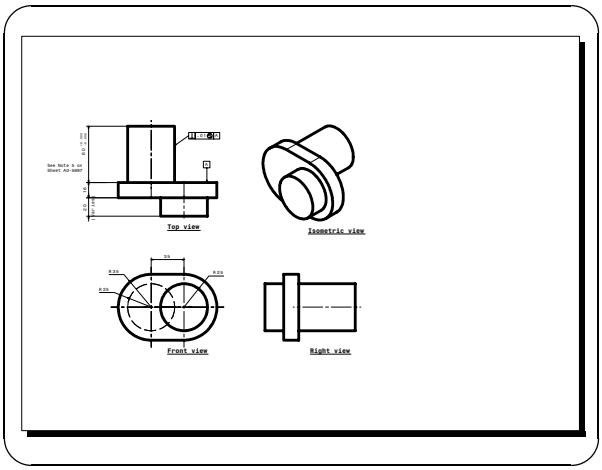
   Select (using **MB3**) the **20**mm dimension, and select **Properties**
   Select the **Dimension Texts** tab and enter ***(Varies)*** below the Main Value
   Select **OK**

6. Drawing Generation *(continued)*

9. Add annotation



10. Save the Crankshaft drawing



9. **Double-click** the **Top View** (in the specification tree) to underline this view as the **current/active** view to receive new geometry

Select **Text:** and select a location
Enter 2 lines of text in the **Text Editor** window
Select **OK**
Optionally relocate the text, and select and relocate the 60mm dimension

Note: *The extent of* ***Dimension Leader Lines*** *can be* ***individually adjusted*** *by first selecting the dimension, and then* ***Ctrl-selecting*** *and dragging the square symbol at the end of the leader line to be modified. Alternatively,* ***double-click*** *the square symbol to enable a numeric value for the Blanking to be specified.*

Select **Geometrical Tolerance:** (located with the **Datum Feature:** icon)
Select the vertical line as **reference geometry**

Note: *Optionally hold the* ***Ctrl*** *key to create* ***vertical text****...*

Select a **location**
Change the **Tolerance Feature modifier** symbol to
Enter a **Tolerance Value** of **.01**
Optionally insert a Tolerance Value symbol
Enter ***A*** as the Datum Element Reference character, and then select **OK**

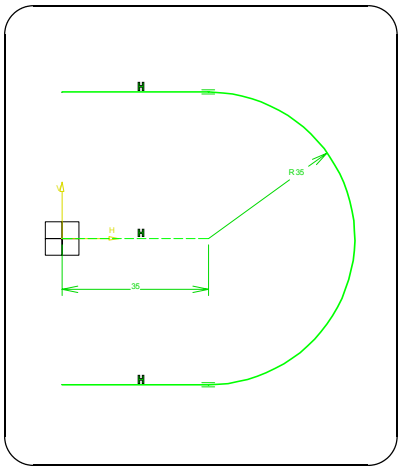
Select **Datum Feature:**, select the **reference** line, the **location**, and select **OK**

10. Select **File + Save As...** to save the CATDrawing as ***Crankshaft_dwg***

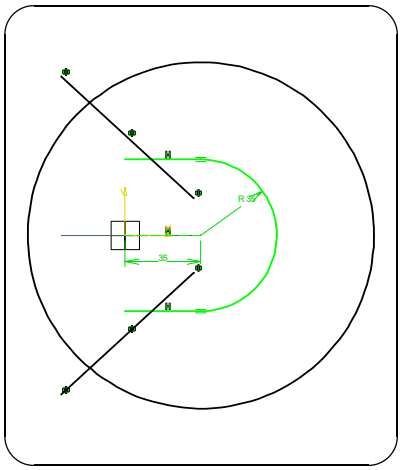
Overview Example

7. Simple Modifications

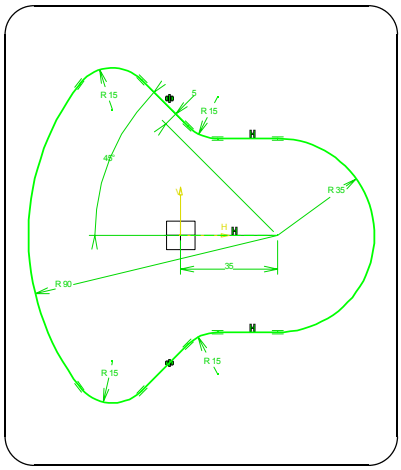
1. Edit existing geometry



2. Create new geometry



3. Add fillets and dimensions



1. Select **Window** + **1. Crankshaft.CATPart** (which activates the **Part Design** workbench)

   Select **Sketch.1** (from the specification tree) and then **Sketcher:** (or vice-versa)

   Select and **Delete** the left-hand semicircle and the vertical dimension
   Double-click Constraint: and create the dimension for the right-hand semicircle
   Select each horizontal line and use **MB3** to swap the dimension for **Horizontal**

2. Select **Circle:** to create a circle **concentric** with the right-hand semicircle

   Select **Line:** to create a line above the common centre point **at approximately 45°**
   Select **Symmetry:** and the **line** (of symmetry) at the axis to mirror the angled line

3. **Double-click Corner:** to define the 4 fillet curves approximately as shown
   Double-click each radius dimension **value**, and enter the required value of **15**mm

   Double-click Constraint: to dimension the circle centre **point** to the angled **line**
   Create an angle dimension from the horizontal **line** at the axis to the angled **line**
   Create a radius dimension to the left-hand circular arc

   Double-click each dimension value, and enter the required values, as shown
   *(Left hand arc radius **90**mm, line to be at **45°** and offset by **5**mm from semicircle centre)*

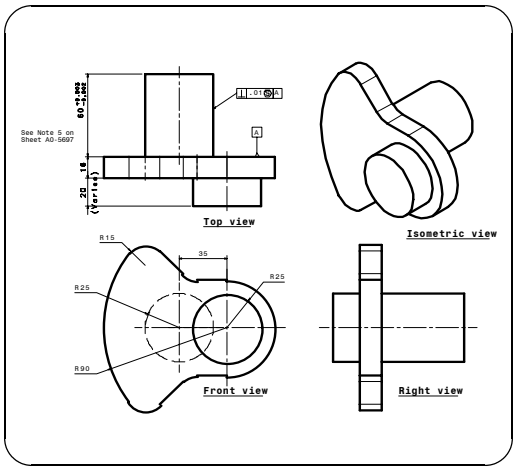
   Select **Exit:** and **note** that the **solid is updated** to take account of the new profile

   Select **File** + **Close** and **Yes** to save the changes made to the ***Crankshaft*** Part

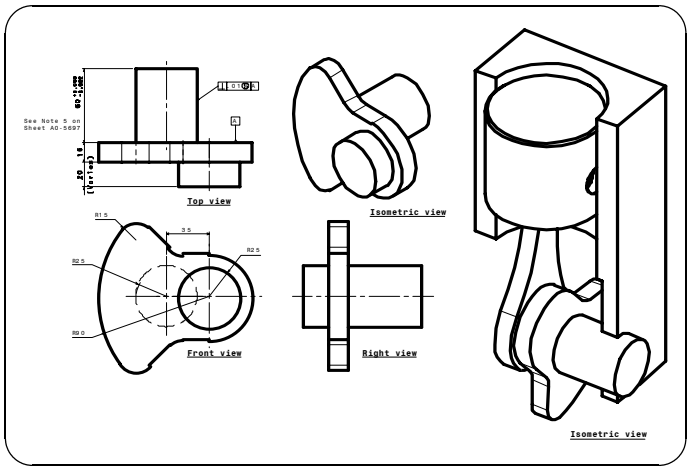
   *(The Crankshaft **Drawing** becomes the current document, and the **Drafting** workbench is automatically activated)*

7. Simple Modifications *(continued)*

4. Update and add to existing drawing



5. Add isometric view displaying Part in assembly



4. ***Note*** *that, following a modification, the affected icons in the specification tree in the Drawing will indicate that an update is required...*
   Select **Update:** which updates the view geometry

   Select and relocate the existing annotation (and **delete** any invalid dimensions)

   Optionally select **Dimension:** to define additional annotation...
   For example, when creating a ***45°*** *angle dimension between 2 selected lines...*
   MB3 can change a proposed type of dimension to be created (from Distance to **Angle**)
   Similarly **MB3** can change the **Angle sector** to generate **45°** (rather than 135°)

   Note: *Holding* ***Shift-Ctrl*** *can also be used to switch the proposed angle between 45 and 135° as the cursor is moved between quadrants*

5. Select **Open:** and **Engine_assy**
   Note *that the modifications to the Crankshaft Part are automatically reflected*

   Select **Window + Tile Horizontally** to **display both** Engine_assy and Crankshaft **windows**
   Select the **Crankshaft_dwg** Document window
   Select **Isometric View:** , the **Block** Part, and then the proposed view
   Remove the view **frame**, and change the view title **text** to match the existing view texts

   Select the window icon to **close** the ***Engine_assy*** and ***Crankshaft_dwg*** documents, selecting **Yes** in response to the *"**Close** - Do you want to save the changes you made to Crankshaft_dwg?"* warning message

Overview Example

(Review...)

★ This example has introduced the **Part Design**, **Assembly Design** and **Drafting** modules with 3 simplified engine components, which were assembled, animated and then modified to show how a drawing sheet of views reflects changes to the master 3D geometry.

★ You will have noticed that procedures are explained from first principles, illustrating both good methodology and also the assumptions made by the modeling process.

★ If you have previously encountered official explanations, such as...
"The resulting geometry is backward the generative one"
"This method realize a breakout view on the view given as parameter"
or *"It is welcome to catch an entity very precisely"*
then you should already appreciate the clear concise English of a CAD/CAM Partnership User Guide, which uniquely compliments and clarifies the on-line documentation provided as standard.

Meanwhile some overview pages which follow (extracted from the **CATIA V5 Introduction User Guide**), give you some idea of the additonal examples...

★ A suspension bracket example introduces 'Bodies', the 'Specification tree' and the use of 'Boolean' and fillet operations to define complex topology.

A handbrake model is constructed to illustrate parameterised sketching and the use of variable fillets and the shell operation in the definition of thin plate parts.

★ The original engine assembly is extended to illustrate the duplication of Parts within a master Part and of components within an assembly, the effects of modifications, and the concept and management of 'External references'.

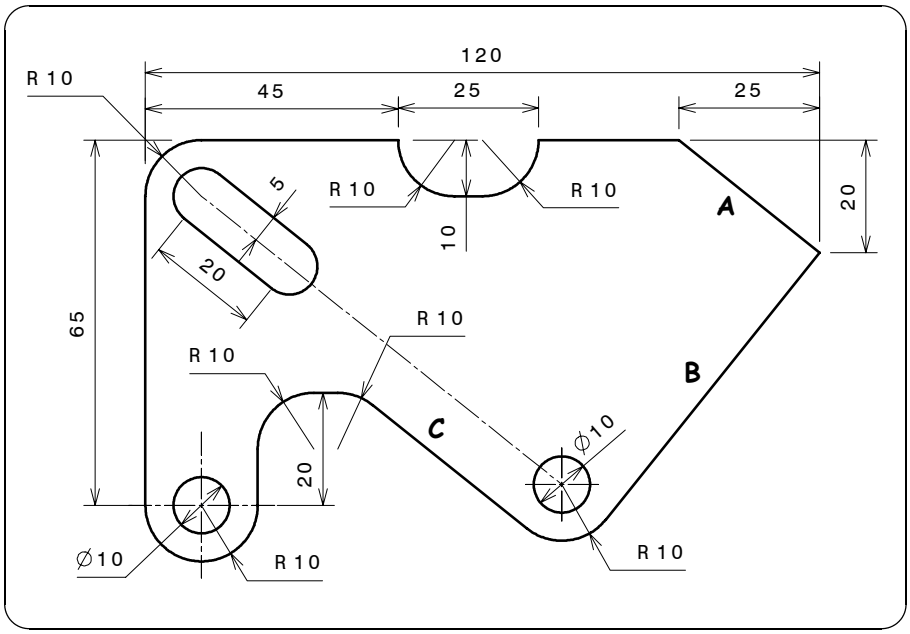
Other examples introduce Patterns, using Catalogs and working with Version 4 data.

★ The complete **CATIA V5 Introduction User Guide** forms the basis for a proven and effective **5-day** course. In fact, instead of merely providing nominal course material, the illustrated worked examples format of CAD/CAM Partnership User Guides is specifically intended to be a useful source for future reference, which is vital considering the vast range of functions and options provided by CATIA V5, and that not everybody will use CATIA on a daily basis.

★ If you are interested in learning more about CATIA V5, then an overview of some of the other CAD/CAM Partnership CATIA User Guides is also included.

3. Sketcher Profiles

Plate Profile



Notes: Lines **A** and **B** are **perpendicular**
Lines **A** and **C**, and the sides of the 20mm **slot**, are **parallel**
The 10mm diameter **holes** are **concentric** with the outer profile fillet curves
The centre of the lower left hole is to be the datum, i.e. a **fixed** point

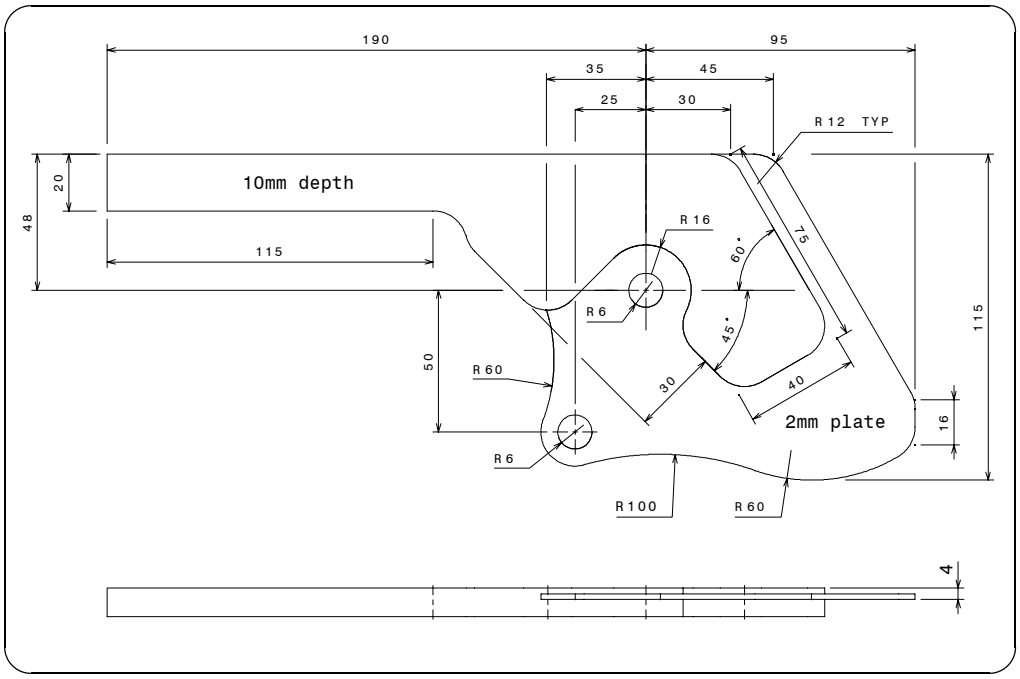
Approach: Use the **Profile:** option to create the basic shape out of **line** segments. Although the Profile option can also create integral fillet curves, it is much easier and more efficient to add fillets later and thereby also implicitly have defined all the required relimitation, tangency and dimensional constraints. The Profile option is also more efficient than the Line option in creating a contiguous sequence of lines.

Both the slot and hole features will be created as **inner profiles**, since their definition must include reference to the outer profile, and they are all through holes.

Comments: The Sketch with dimensional and geometric constraints approach adopted by CATIA Version 5 may sometimes appear more involved than simple (isolated) trimmed lines and curves. However, any subsequent **modifications** can be infinitely **more simple and powerful**.

Sketcher Profiles

Handbrake Profiles



Notes: All radii are **12**mm unless shown otherwise.
The **75** and **40**mm distance dimensions are perpendicular.
There are two profiles which share some common geometry.
Should the main (10mm depth) profile subsequently be modified, then it is expected that the secondary (2mm plate) profile should adjust accordingly.

Approach: The **Profile:** option will again be used to create the basic shape out of line segments, but will also incorporate the **Tangent Arc:** and the (non-tangent) **Three Point Arc:** options as required.

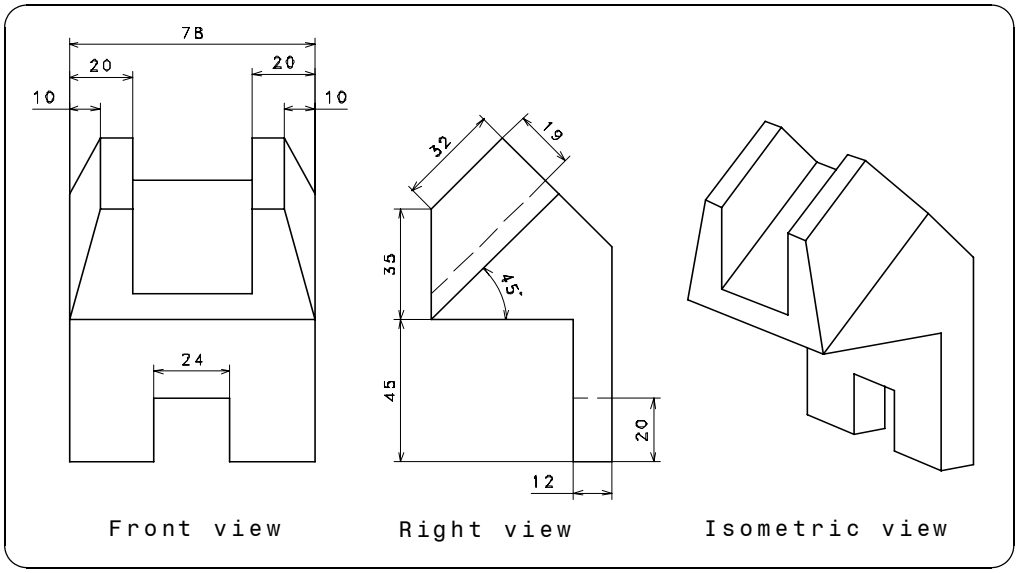
In this example, as many of the **known dimensions** as possible **will be specified as the profile is constructed**.

Note that the creation of a new '**Body**': is required to logically separate the solid geometry of the 2mm plate - to facilitate subsequent additions to the main (10mm thick) solid Body, including a 'Shell' operation which must not include (or be affected by) the 2mm plate solid.

Comments: The ability to **specify known values**, such as coordinates, a length, a radius and/or an angle, **during the creation of a profile** in the Sketcher is **very powerful**.

4. Prismatic Parts

1. Planar Support Bracket



Notes: The specified dimensions will be used, where possible, to constrain and dimension the geometry. These same dimensions should then be generated by the Generative Drafting module if automatic dimensioning of the views from the part is requested.

The 24x20mm cutout is assumed to be centred within the face in the front view.

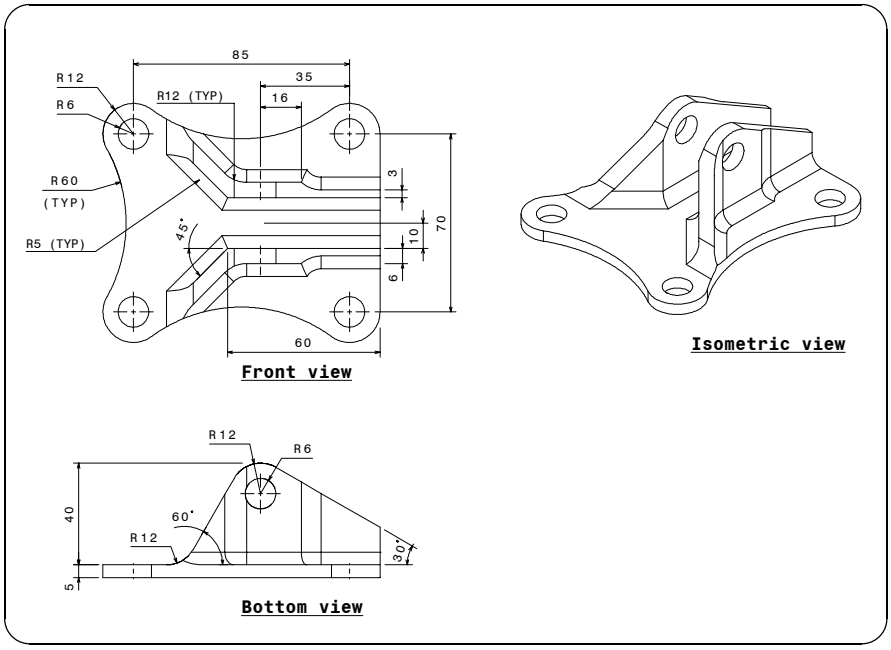
Approach: The Part is largely defined by its side elevation, which can define a **Pad:** 

The top cutouts will be defined in the rear 45° face plane, using **Project 3D elements:**  to create reference geometry from existing edges.

Comments: Prismatic parts are particularly suited for modeling in solids. In this simple example, it is unnecessary to consider logically grouping the features in the specification tree.

Powerful Part modification options are available due to the implicitly intelligent model resulting from the use of **reference elements** and the specification of **Pocket:**  limits.

2. Suspension Bracket



Objective: To illustrate how quite complex topology can be simply created by using the 'Boolean' operations - particularly the **intersection** of two solids.

Since many sketch profile dimensions are provided, the opportunity is also taken to further evaluate the effectiveness of automatically generating dimensions.

Approach: Here again the creation of a new 'Body' is required to logically separate the solid geometry as it is created - only then is it possible to **specify each of both solids** (to be added, subtracted, or intersected), otherwise, by default, the selected solid will be Booleaned only with the solid of the PartBody.

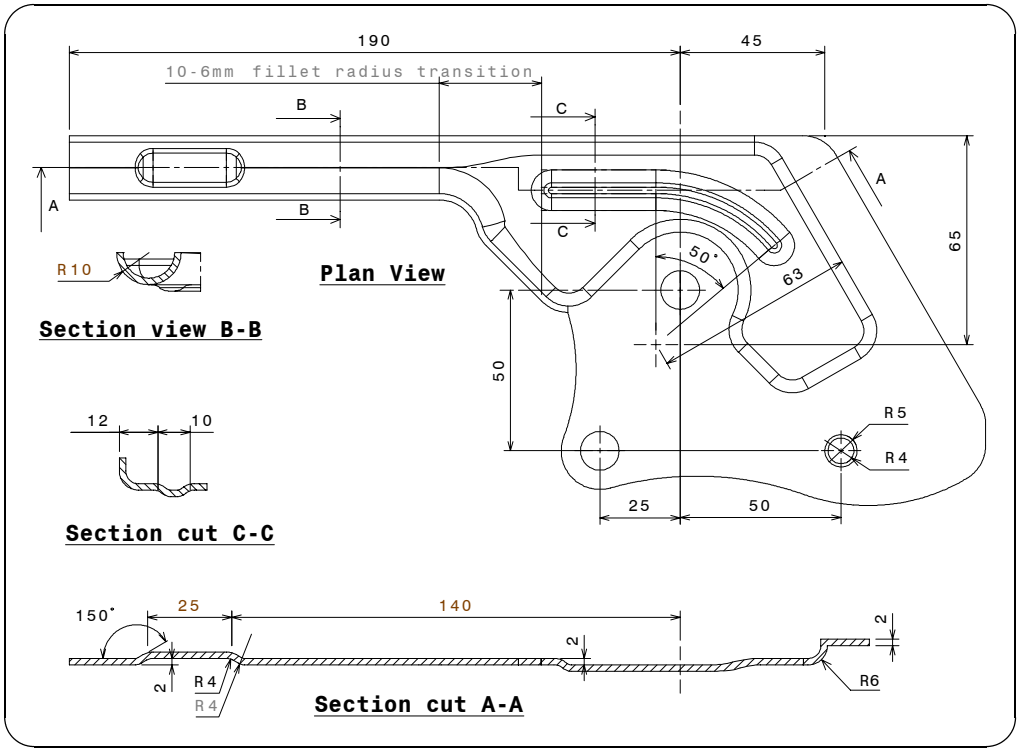
A **'cut and paste'** methodology is introduced to as an alternative method of defining (initially) identical dimensional constraint **parameter values**.

Comments: As with all solid modeling software, careful consideration has to be given to the sequence in which the geometry is generated and how it is combined.

The Fillet function copes well where the geometry extends to the edges of the base plate, although a warning message may be displayed.

It is still more productive to manually define individual annotation dimensions, of the required format, in the appropriate view, and at the most aesthetic and clear location, rather than use the options which automatically generate dimensions.

3. Handbrake Plate



Objective: Primarily to illustrate the use of the **Shell:** operation in the modeling of **thin plate parts**, which typically incorporate features such as swages, as shown by the sections above.

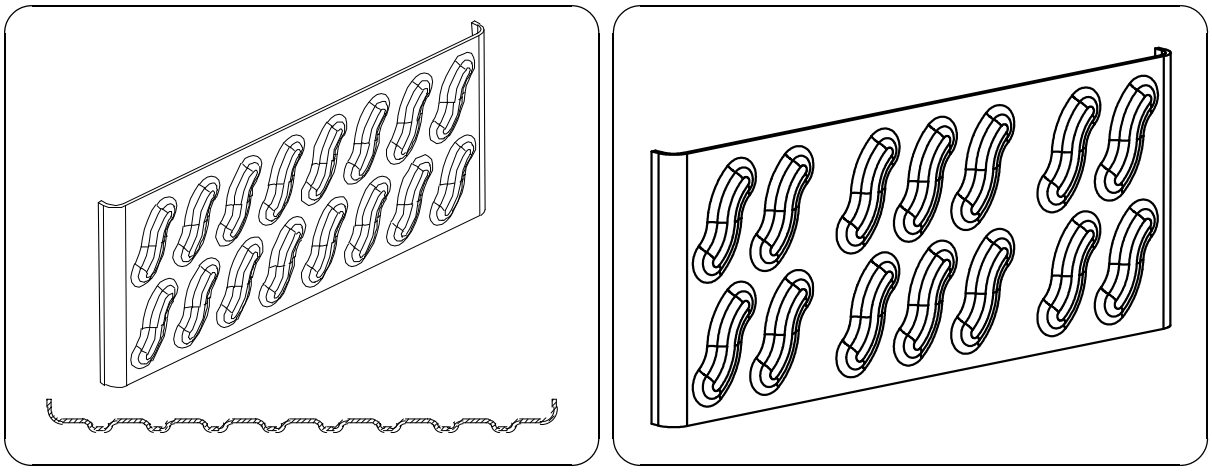
Approach: The complete outer form of the plate is modeled - leaving the Shell operation to the last possible moment. This approach can generate quite complex thin plate sections, especially if the incorporated features are filleted.

A **Variable Radius Fillet:** will be used to fillet the 10mm plate profile, starting and ending with R10, and blending with R6 to accommodate the 2mm thick plate profile.

The **Union Trim** operation is also introduced to illustrate how a more advanced 'Boolean' can both add and subtract solid at the same time, thereby producing, for example, a 'boss **and** a through hole' as a single feature.
(Note that a Union Trim may not be available within a P1 software configuration)

Comments: Formed plates might not immediately be considered as 'prismatic parts', but planned use of the powerful Shell operation can efficiently generate quite complex geometry.

4. Patterns of Objects



Objective: To generate a tread pattern on a plate to illustrate the use of **Rectangular Pattern:** and **User Pattern:** to generate regular and varying repetitions of specified geometry.

Approach: Advantage is taken of the symmetry within the tread geometry to illustrate how a profile can be almost completely defined by geometrical Constraints.

Rectangular Pattern: is used to define a regular 30x50mm grid of the tread geometry, referencing the orthogonal sides of the plate to define the directions.

The plate with the pattern of treads is 'shelled' with a thickness of 2mm.

The Rectangular Pattern is then replaced by a **User Pattern:**, which requires a Sketch of Points to define the instances of the thread geometry.

Comments: Should the Pattern command icon be activated before the selection of geometry, (typically the methodolgy elsewhere), then options may be restricted.

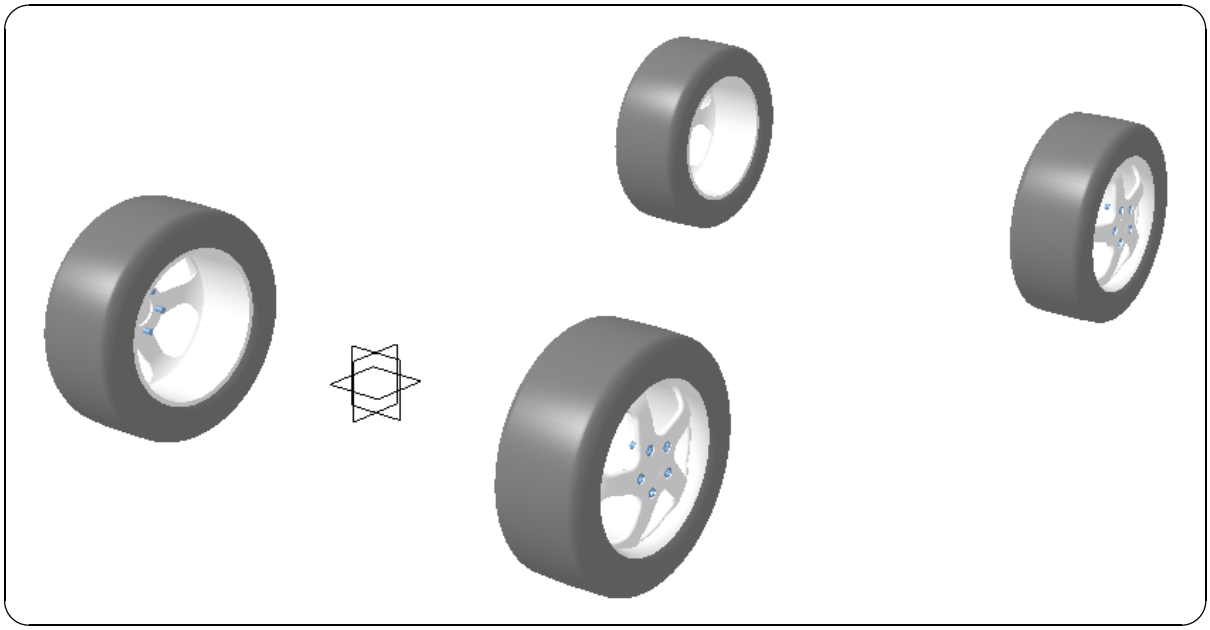
It is important to consider the 'anchor point' of the geometry to be duplicated, since the default may not be appropriate.

For a **User Pattern:**, then care must also be taken not to superimpose a copy of the geometry being patterned at its original location.

Note that a select **Circular Pattern:** option is also available.

Chapter 4

5. Sports Car Wheel



Objective: To create simple models of sports car wheels to illustrate the further the application of Patterns and the retrieval of standard Parts from the sample Catalogs, and to provide an example of Sub-assemblies of Parts in turn used as components within a main assembly document.

Comments: The datum/origin of a Hole based on a non-planar face may not obviously relate to that of the Part, and the orientation of the axis of a hole defaults to be normal to the supporting face, (which is not appropriate in this example and will need to be modified).

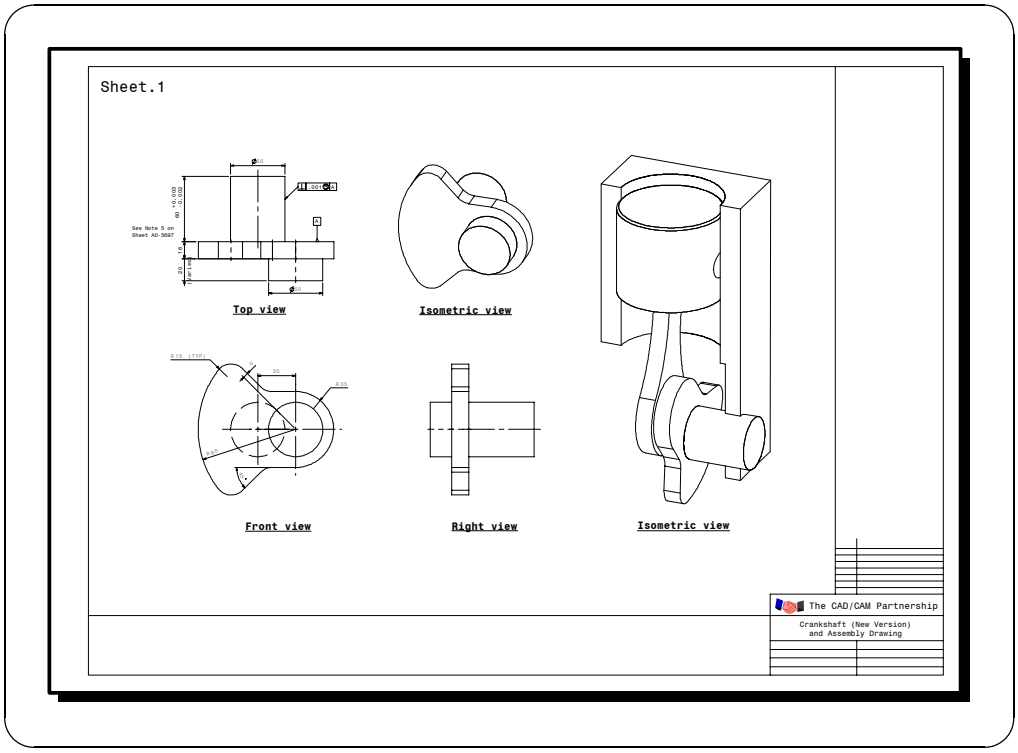
Use of the standard Catalogs requires that the document search order settings include the search of all sub-directories, in addition to the specific search of 'Catalog & startup documents' option.

Care needs to be taken when a Component is 'mirrored'. In this example, an opposing instance of a wheel is of identical Part geometry, and not 'handed' geometry (which would have to be manufactured separately). Therefore the wheel geometry is instead rotated about a vertical line of symmetry. This clarification would also be apparent in a bill of materials/list of components used.

Both **Snap:** and the parameters window of the **Compass** provide alternative methods to accurately reposition Components within an assembly, and are particularly efficient where a permanent record of Constraints is not required.

5. Draughting and Plotting

Draughting Basics



Objective: To introduce the **Background** and **Working Views** modes which are used to keep the **Sheet frame/title block** geometry separate from that of generated views.

To insert a **company logo** image into the title block, and to create a **new sheet**, within the same CATDrawing document, using the same frame/title block.

To define a **Section View**.

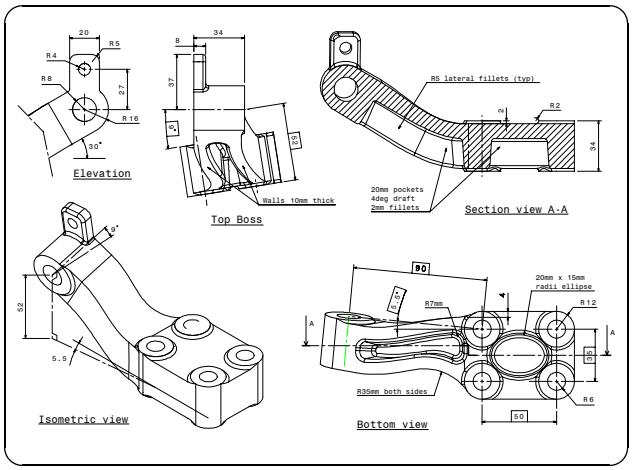
To introduce ***'Draw Details'***, which are first created in a **Detail Sheet**, and can then be repeated as a ***'Dittos'*** on a Drawing Sheet.

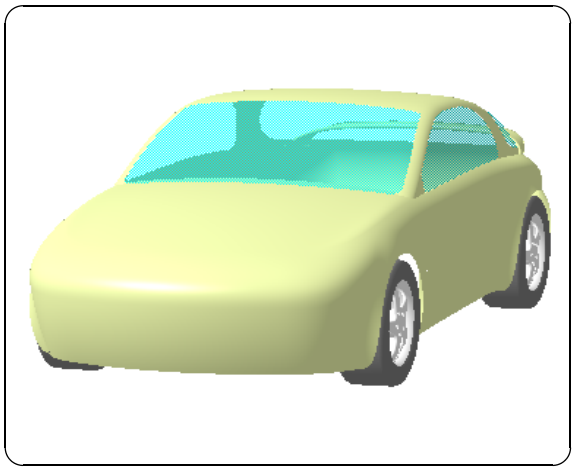
To provide an overview of **plotting** concepts.

Comment: Draw Dittos can not be referenced as geometry, by a dimension for example, and are therefore of limited use.

Sample automatic sheet frame and title block programs are provided by default, as specified by the **Tools + Options... + Mechanical Design + Drafting Layout** tab, e.g. the *E:\CatiaV5\intel_a\VBScript\frameTitleBlock* directory.

8. The CATIA V5 Advanced User Guide





★ Following on from this CATIA V5 Introduction, the CATIA V5 Advanced User Guide provides additional genuine independent insight and easy to follow step-by-step illustrated worked examples, and also forms the basis for a **5-day** course.

★ Describes more advanced Sketch, Part Design, and Drafting concepts, and introduces new topics, such as Assembly Analysis, Interference and Sectioning, Bills of Materials, Surfaces, Layers, Formulas and Design Tables, and 'PowerCopies'.

The main examples include the generation and modification of a Pressure Chest Casing assembly, a complex Engine Bearing Arm which includes twisting and drafted pockets, and the surfacing and modification of a simplified concept car body, as illustrated above.

★ **The CAD/CAM Partnership CATIA V5 Advanced User Guide** answers, for example...

What happens when the Sketch Support Plane is modified ?

How are Drawing Number texts created (for example, both in the title block and also top left of the sheet) so that they remain identical ?

How can I create an exploded view of an assembly which will automatically reflect changes to the constituent Components ?

What are the procedures to create relationships between dimension values ?

How can I place an Excel spreadsheet on a drawing to redefine multiple dimensions and other values, according to a specified member of a family of parts ?

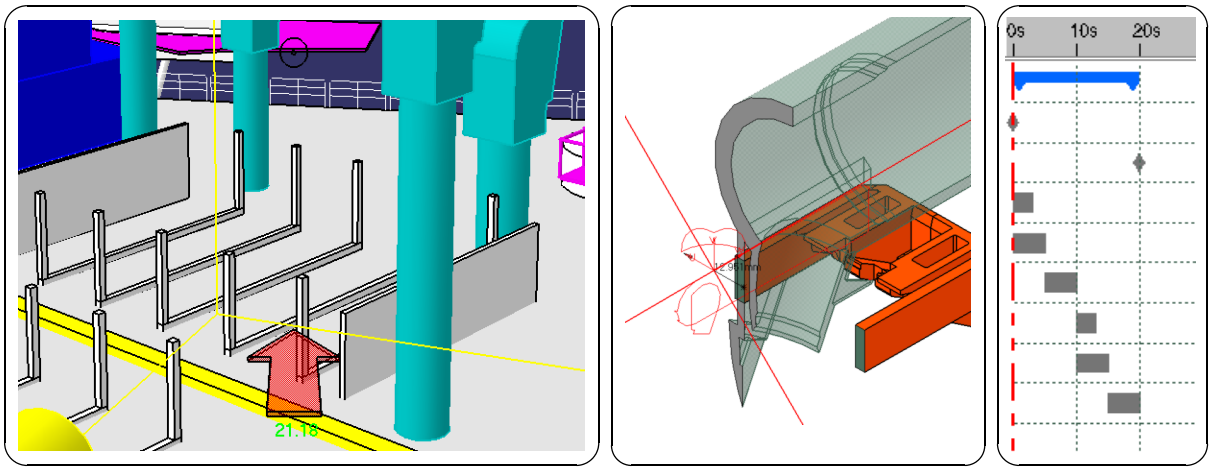
What are 'DLNames' and what is the purpose of file Search Orders ?

How do I dynamically review a designated cross-sectional area of a Part, while reducing the pocket wall thickness to meet a specified weight limit ?

What are Catalog Text Templates, and how can they be used to annotate drawings ?

7. Miscellaneous

9. The CATIA V5 Digital Mockup User Guide



★ This User Guide forms the basis for a **2-day** course and assumes no prior knowledge.

Standard sample data is used to illustrate the definition and display of existing components within a product structure, and the different modes of navigation.

★ The **DMU Optimizer** can further improve the display response, and so examples are provided of the Silhouette: , Wrapping: and Simplification: options.

★ The **DMU Navigator** is used to add 2D and 3D annotation, and to create 'Scenes' of an alternative configuration, such as an exploded presentation.

Examples are also provided of Spatial Queries and Measurements, the comparison of drawings within the 2D Workshop, and also the generation of animated presentations ('Replays' and video files) via Camera Tracks.

★ **DMU Space Analysis** is used to illustrate the Distance and Band (proximity) Analysis of components, the combination of Sectioning and Interference Analysis, and the visual comparison of different versions of (3D) components.

★ The chapter explaining the **DMU Fitting Simulator** provides examples of creating and modifying the assembly paths ('Tracks') of components, creating a 'Shuttle', and how to edit and subsequently annotate a 'Sequence'.

Other examples illustrate Clash Detection during the simulation of an assembly, displaying a Gantt chart, and how to define an 'Experiment'.

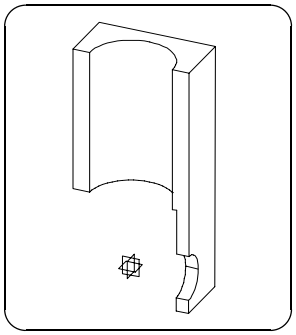
★ The principles of the **DMU Kinematics Simulator** are explained, together with a table of Joint types and their 'Degrees of Freedom'. Examples illustrate the creation of Joints via Constraints, and the definition of Command Laws and kinematic Analysis.

8. Summary

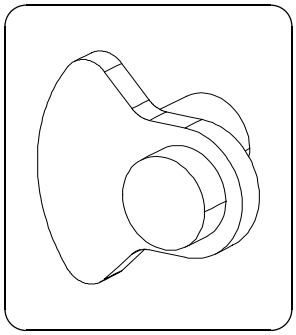
2. A Review of the Examples

At this stage, your working directory, (for example, *E:\Catadata\My_work),* should contain at least 2 CATDrawing documents, and the following CATPart and CATProduct documents...

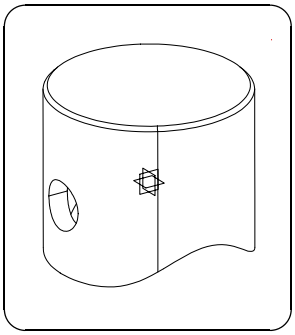
Block.CATPart



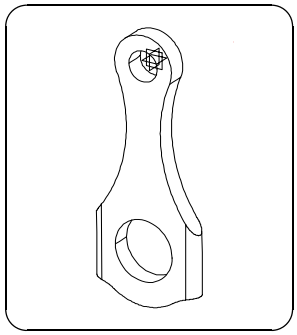
Crankshaft.CATPart



Piston.CATPart



Conrod.CATPart



Engine_assy.CATProduct

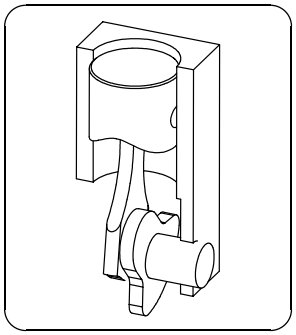
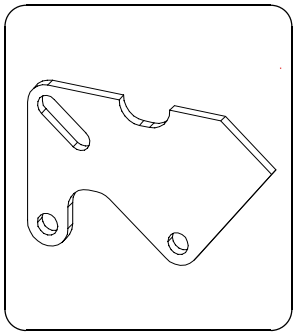
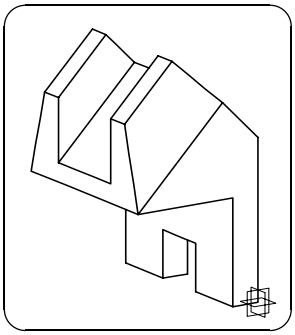


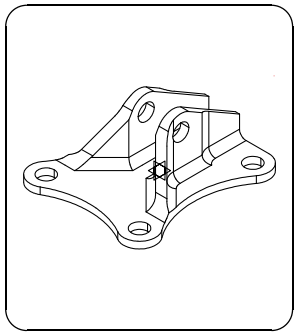
Plate.CATPart



Planar_Bracket.CATPart



Suspension_Bracket.C



Handbrake.CATPart

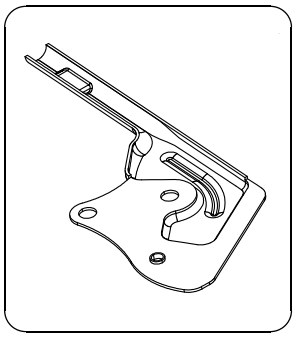
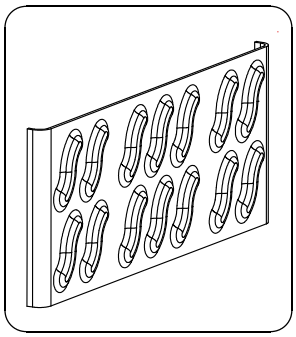
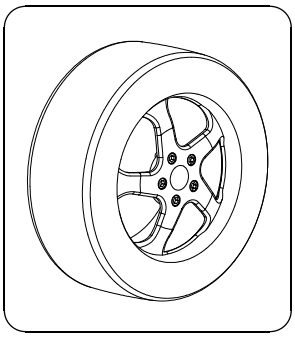
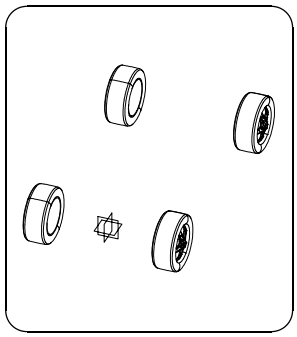


Plate2.CATPart

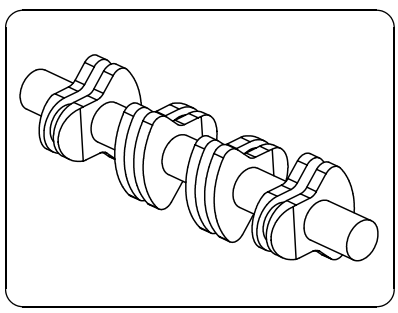


Car_wheel_assy and Car_wheels CATProducts

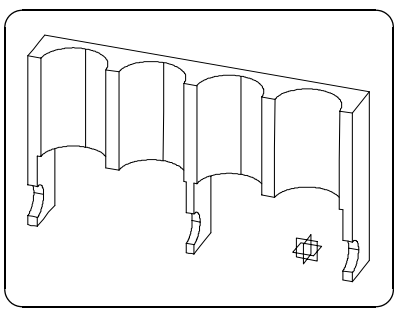




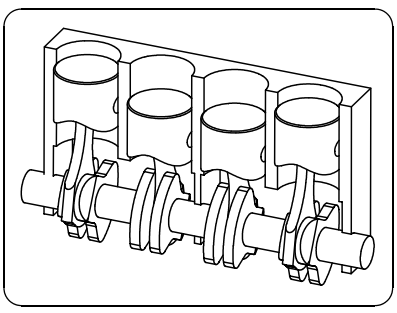
Crankbay/Crankshaft4.CATPart



Block4.CATPart



Engine_assy4.CATProduct



Summary

2. A Review of Examples (continued)

Page	**Example**	**Introduces/illustrates...**
2.1	Block, Crankshaft, Piston and Conrod	Simple Sketches and Part creation Modifying a Part
2.10	Engine_assy	Assembly of Components and defining Constraints
6.5		Creating and managing External References
2.13	Crankshaft_dwg	Creating a new drawing sheet and defining views Defining dimensions and adding annotation
5.2		Creating a sheet frame and title block with a company logo
3.1	Plate	Defining geometrical constraints within a Sketch
4.1	Planar_Bracket	Creating a prismatic solid with non-orthogonal faces
4.7	Suspension_Bracket	Creating a NewBody and managing the Specification tree 'Boolean' operations, particularly the use of Intersect Defining fillets, and 'automatic dimensioning' of draw views
3.7	Handbrake	Defining known dimensions as the Sketch is constructed Variable radius Fillets, and the Union Trim operation
4.15		Using the Shell operation to define thin plate parts The concept of the In Work Object
4.24	Plate2	Rectangular and user defined Patterns of objects
4.30	Car_wheel	Circular Patterns, and Holes in non-planar faces
4.33	Car_wheel_assy	Retrieving a standard Part from a Catalog Positioning a Part in an assembly with respect to mating Parts
4.34	Car_wheels	Moving a (sub-assembly) component within an assembly Creating symmetrical instances of components within an assembly
6.10	Crankshaft4	Duplicating Parts within a master Part
6.12	Engine_assy4	Duplicating Components within an assembly Changing referenced Parts

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Feedback

As part of our commitment to the quality and effectiveness of our **CATIA V5 User Guides**, we welcome any queries or comments that you may wish to email to...

info@cadcam.co.uk