

CATIA V5 R12

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Feb 2004

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1. Introduction

A cautionary note before you begin to use CatiaV5. Catia can sometimes freeze the system on some computers, it can also crash if you are unlucky, so it is advisable to save your work regularly during work sessions.

The CATIA V5 package consists of modules which contain various workbenches (eg *Part Design*, under *Mechanical Design* in *Start* menu) containing icons which access menu operations. The most commonly used workbenches are the *Part Design* workbench for creating solids, *Generative Shape Design* workbench (under *Shape* in *Start* menu) for creating so called open bodies which are surfaces and *Assembly Design* workbench to assemble part bodies and open bodies. Solids and open bodies are saved as *CATIAParts* and assemblies are saved as *CATIAProducts*. When creating your design it is best to arrange it to be an assembly of relatively simple parts. You can then open the product and work on each part in the context of the assembly. Hence you can create wing rib parts from a wing shape where the wing surface is one part and the rib another part in the same assembly. The kinematic example in this tutorial shows how to assemble parts using the *DMU Kinematics* workbench. This is very similar to using the *Assembly Design* workbench..

Start CatiaV5 by double clicking the shortcut – wait for it to start, this can take a minute or so. The first time you do this you will probably get a window requiring you to select the licence required. Select the box and OK, then restart CATIA. If you save and recover your settings as in the next section you will not have to do this again. Having started CATIA, check out the available modules by selecting *Start* and drag to each module. You will then see the workbenches under each module. For example Mechanical Design module has workbenches Part Design, Assembly Design, Sketcher, Structure Design, Drafting, Sheet Metal Design and Wireframe and Surface Design. CATIA V5 provides extensive online help with tutorial exercises as well as explanations of operational procedures. There are various ways of selecting help, each gives you different types of information. The following sections will lead you through the help facility and give you a start. There are then some examples of part design, surface design and machining.

Before you start here is a brief description of how to use the catia interface.

You will be asked to select menu icons. These are usually on the right vertical menu. Place your cursor on each icon in the menu to get a description of its function. If you can't find an icon it is sometimes easier to use *Insert* on the top menu and find it there. Always use the left button for these selections.

To manipulate your model in 3D use the mouse.

To pan the model, press the middle button and drag.

To rotate, press the middle button and hold followed by holding down the left button – drag while holding both buttons down.

To zoom, press the middle button and hold and click the left button – drag up and down.

Assignment submission

Catia assignment submission requires you to create a folder called *your student number* which contains all the Catia files required. To make sure you include all the files necessary, create the folder then use the Catia command *File/SendTo-directory*, select all the files listed and the folder you created. This will ensure that all the files necessary for your assignment are submitted.

2. Personal Settings and Customisation

CATIA allows you to personalise and customise settings. These control what the menus look like, what is visible by default in the structure tree, what favourite workbenches you like to run etc. These are set by selecting in CATIA *Tools/Options* or *Tools/Customise*. When following some of the tutorials, you are instructed to change some of these settings.

All this information is stored in *Winnt\Profiles\username\Application Data\DassaultSystemes\CATSettings* and *same path\CATEnv* (both of which are directories).

If the computer you are using is for general use, CATSettings and CATEnv will be lost when you log out. In order to be able to start CATIA next time with all the current settings you must copy them to a personal place (eg. Z:)before you log off. Then, the next time you log on, copy them back before starting CATIA. If you do not do this you will end up with default values instead.

A useful tip I have found – if your menu icon placements get in a mess and you want to keep other settings, you can reset the menu placements to default by deleting the file *DialogPosition* in your CATSettings directory.

3. CATIA V5 Infrastructure

If necessary start CATIA using the shortcut.

Select *Help/Contents, Index and Search*

The first time you select Help you may get a window asking you to specify the path for the help files (this is saved in your personal settings).

Select the path *C:\Program Files\Dassault Systemes\B07doc*.

This starts your browser which is used for all the online help. You can switch between the CATIA session and the help session using the usual windows methods.

In the help browser, select *Infrastructure/CATIA Infrastructure Users Guide*

The left column contains links to;

- *Preface* (the current page)
- *What's New*
- *Getting started*
- *Basic tasks*
- *Advanced tasks*
- *Workbench description*
- *Customising*
- *Installation requirements*
- *Glossary*
- *Index*

This format is very similar for any of the selected help topics and can be used to go further into the help.

Check out *Preface*, *Basic Tasks* and *Workbench Description* for *Infrastructure* then proceed to the next section.

4. Example – Getting Help on Mechanical and Part Design

If you want to learn more about a particular module and workbench (eg. Mechanical and Part Design), in a CATIA session, select *Help/Contents, Index and Search*.

As mentioned above, the first time you select Help you will need to specify the path for the help files. The path is;

C:\ProgramFiles\DassaultSystemes\B10doc

This will be saved if you save your settings.

In the help browser, select *Mechanical Solutions/Part Design User Guide*.

Select *Basic Tasks* (in the left column)

Select *Sketch based features*

Select *Create a Pad*, go to the top of the document and follow the instructions.

Note, selecting *Pad1.CATPart* will find and get CATIA to open the example file required, starting CatiaV5 if necessary.

You can jump directly to any of the sections in Part Design help using the list on the left (*Preface, What's New, Getting Started, Basic Tasks, Advanced Tasks, Workbench Description, Customising, Glossary, or Index*).

When in a CATIA session, you can get brief help on a workbench function by selecting *Help/What's This?* Place the cursor on the required workbench icon and click to get more information.

You can also get tutorial help on a menu selection by highlighting the menu icon then selecting *Help/CATIA V5 help* This starts the help browser as before and locates a tutorial example or more help on the selected menu.

It is recommended that you become familiar with Help, especially Contexts, Index and Search as it will answer most questions and allow you to become proficient with CATIA much faster.

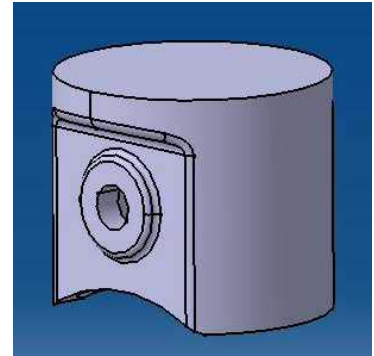
5. Part Design Example – Creating a Piston

For all the following examples it is recommended that you first familiarise yourself with the way CatiaV5 operates by using the help as shown in the previous sections.

Before you start select *Tools/Options/Mechanical Design/Part Design*
And set *Update to Manual*

Before selecting OK select *Mechanical Design/Sketcher*
Set grid spacing of 20 and graduations 100
Uncheck *snap to point* - OK

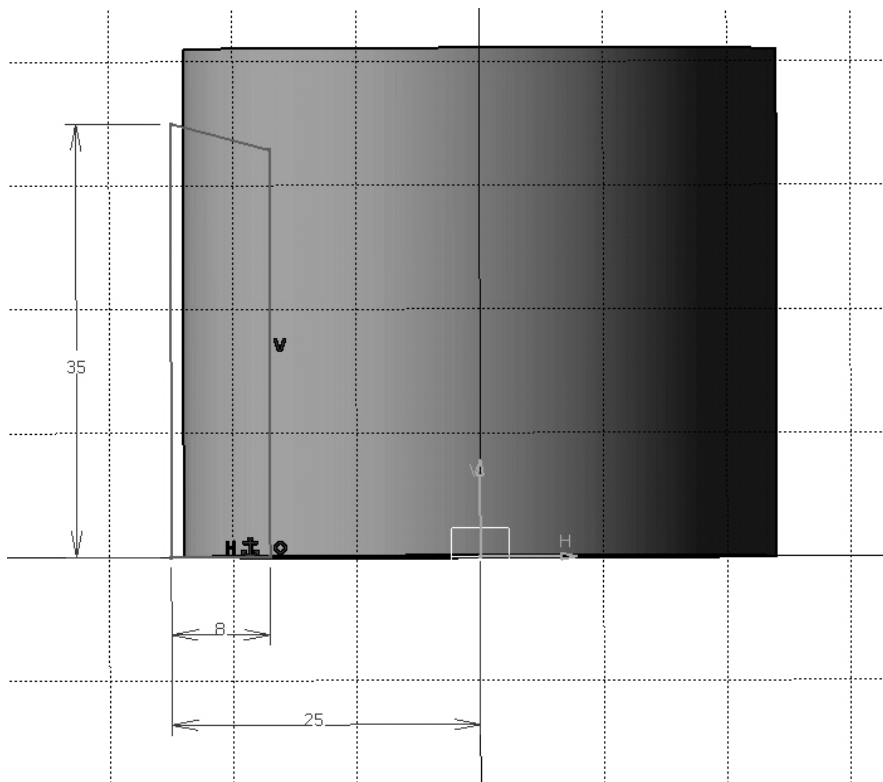
Select *File/New* and choose *part*
We will create a piston as shown.



Select the XY plane then the *sketcher* icon (**Hint:** if at any time you can't find an icon in the menu bar, you can use *Insert* (in the top menu) and find it that way.)

Select *circle* icon and sketch a circle centre at origin, by clicking first the centre point then a circumferential point.
Select *constraint* icon and select the position for the dimension.
Double click the dimension and modify it to 48mm diameter - OK
Exit the sketcher and select *pad* icon (notice the sketch was still active and the pad is automatically referring to this).
Fill in 41mm as pad length - OK.

We will now create a second part body as follows;
Select *Insert* menu, Select *Body*
This creates a separate feature editor branch in the specification tree
Select the ZX plane then *sketcher* icon
Select *profile* and sketch the profile shown.



Double click *constraint* icon (double click locks the command so that it can be re-used)
Place the cursor on the lower line which is on the H axis and right click, then drag to *fix*.

Still with *constraint* active select each separate line and select the position for each dimension – you can double click each dimension to modify it to match the figure on the right.

To create the angle, select the two appropriate lines then select the position for the dimension – double click this to change it to the required dimension.

Repeat this for the distance dimension from the V axis.

Angle=15°, height=35, width=8, dist. from V=25.

Select *exit* icon.

Select *pad* icon (sketch is still active and automatically used)

Select *mirrored extent* - OK

This pad is independent from the original part body, we will mirror this about the YZ plane;

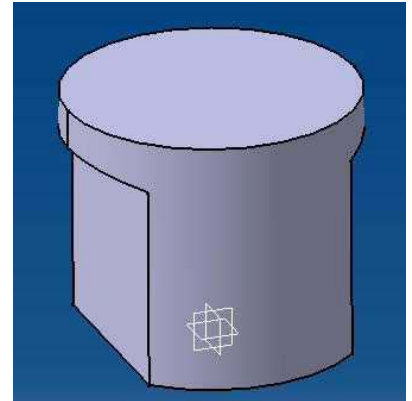
Select *mirror* icon, select the YZ plane - OK

We wish now to remove this body from the original one, to do this place the cursor on *Body.2* in the Feature Editor Tree.

Right click and drag to *Body.2/object/remove*.

OK

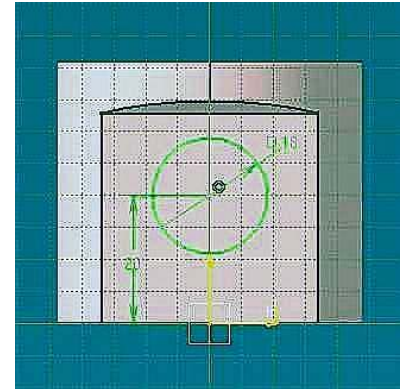
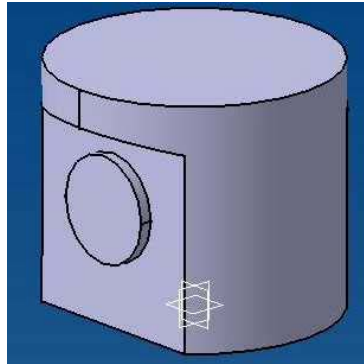
This should give the result on the right.



We now want to create the double boss shown on the right.

Select YZ plane and *sketcher* icon.

Using *circle* icon create a circle as shown on the far right.



Select *constraint* icon and create the constraints;

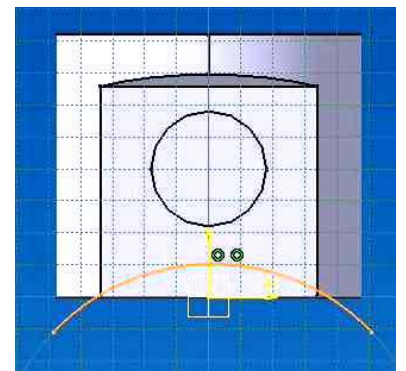
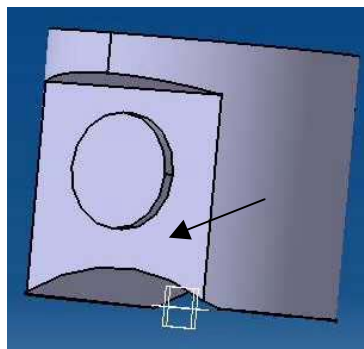
circle diameter = 18, circle centre 20 from bottom edge.

Exit the sketcher

Select *pad* icon.

Select *mirrored extent* and *length* 19mm

OK creates the boss on both sides.



We now want to create the cutout shown on the right. To do this select the arrowed face the select *sketch* icon.

Select *3 Point Arc* icon (this is under the *circle* icon)

Sketch one end point, the mid point then the other end of the arc as shown.

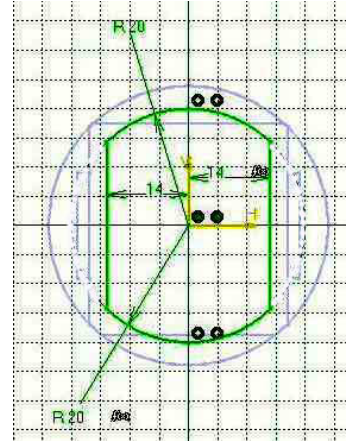
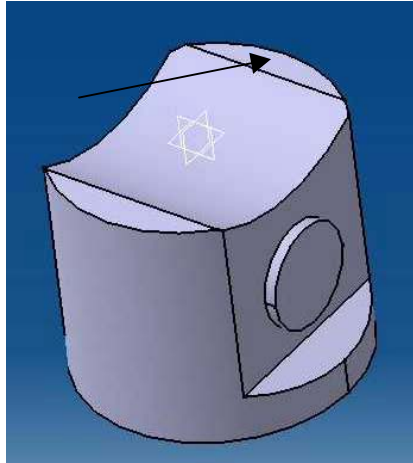
You could constrain this as before but we will leave it unconstrained for this exercise.

Exit the sketcher and select *pocket* icon

If necessary, select the arrow to be pointing downwards, the highlighted area will be removed.

Select *up to next* as the option - OK creates the cutout.

We now want to hollow out the inside of the piston.
Select the face arrowed and *sketcher* icon.



Select *line* icon and draw one vertical line as shown (don't worry if this profile is not exactly the right shape, constraints will fix that up later)

Select *3 point arc* and draw one of the arcs from the end of the line.

Select *line* icon again and draw the next vertical line, followed by the other 3 point arc.

Double click *constraint* icon and create radius dimensions for each arc.

Select the top circular edge of the cylinder, then the top arc sketched. By right clicking now drag to *concentricity*.

Repeat this for the bottom arc.

Double click the top radius dimension and set it to 20mm.

Double click the bottom radius dimension, right click the *radius* field of the dialog box and drag to *Edit formula*.

Select the top radius dimension – OK. This constrains the bottom radius to be the same as the top.

Create the two dimensions between the V axis and the vertical lines of the profile.

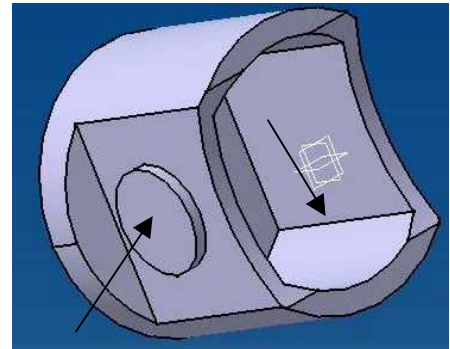
Double click the right side dimension and right click the constraint definition dialog box and drag to *edit formula*.

Select the left dimension - OK - OK

Double click the left dimension and set it to 14mm.

This should give the sketch shape above..

Exit the sketcher.



Select pocket icon

Set *type* to *dimension*

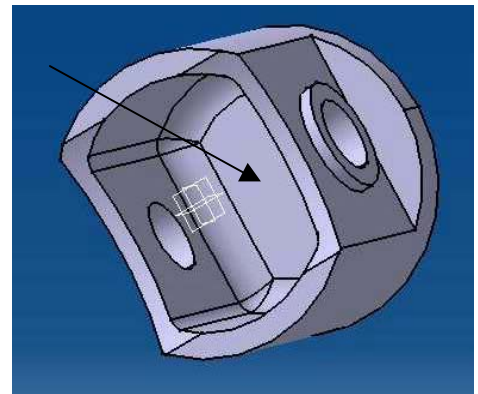
Set the depth to 37mm

OK creates the pocket shown.

To create the through hole select the round face arrowed.

Select hole icon, select up to last, set the diameter to 12mm - OK

(This is automatically concentric with the boss)



Filleting the edges.

Select the internal, arrowed edge above and holding the control key, select the other 3 similar edges as well.

Select *edge fillet* icon.

Set the radius to 3mm - OK

Select the internal face arrowed on the right.

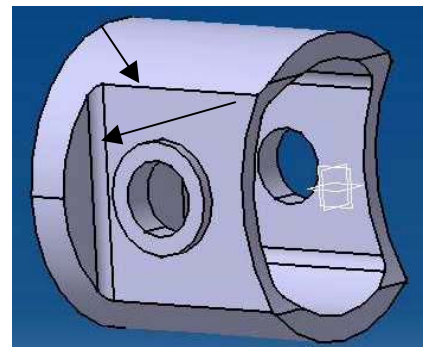
Select *edge fillet*

Set radius to 5mm - OK gives the new fillet on the right

Create the two symmetrical fillets arrowed on the right to be 2mm radius

Then select the arrowed edge and its symmetrical one on the opposite face using the cntrl key.

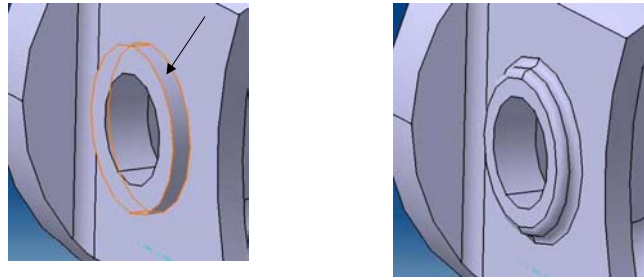
Select edge fillet and set the radius to 2mm - OK.



Select the arrowed edge face and its symmetrical face on the opposite side of the piston then *edge fillet* icon.

Fill in 1mm for radius
OK

This gives the fillet on the far right



Apply material properties to the piston by selecting *Apply material* (on bottom menu). Select *metal* tab then *Aluminium*. Select the piston, *apply material*, OK.

Select in the shading icons *apply customized view parameters* (and if necessary select *materials* in new window) OK

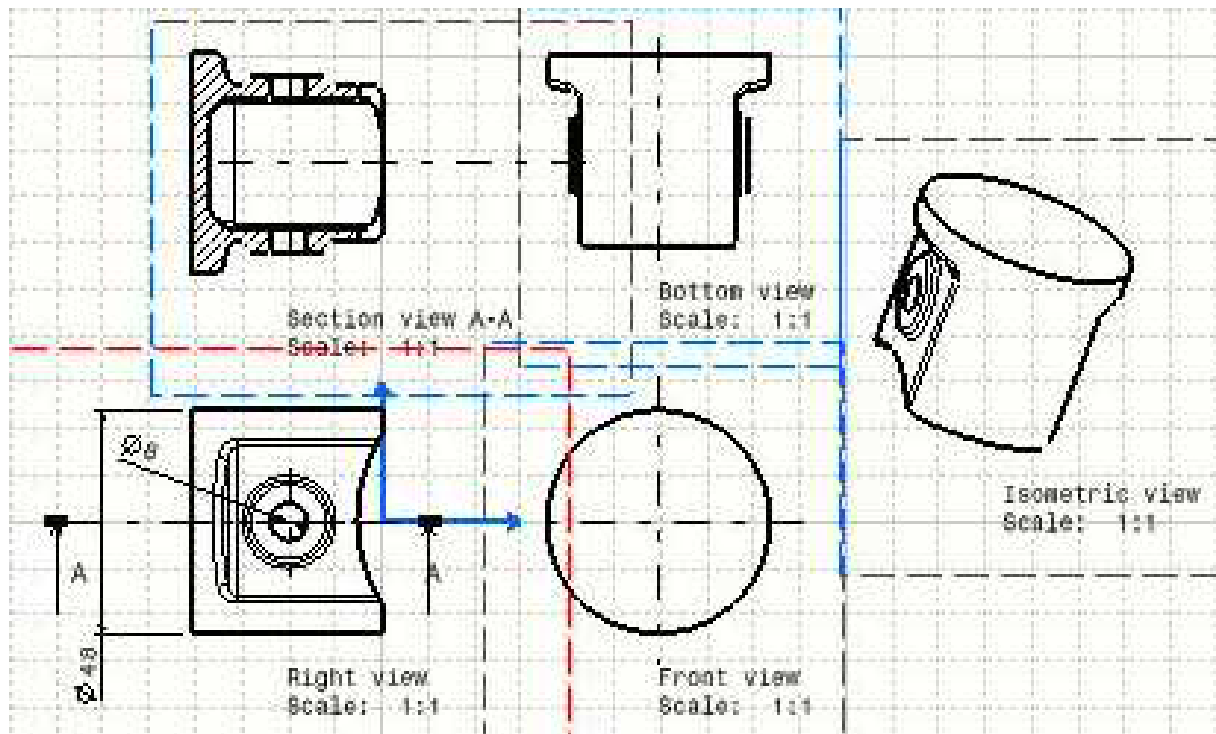
6. Drafting Example – Creating a drawing of a Piston

If you haven't completed the example in section 5 read *C:\CatiaV5_training\UNSW\Piston.CatPart*.

Select *Start* menu, *Mechanical design – Drafting* workbench

In the *new drawing creation* dialog box, set the layout to be *Front, Bottom and Right* (place the mouse over the layouts to find the right one)_

Select *modify* in the same dialog box and select *A3ISO, scale 1:1* - OK creates the drawing views “right”, “bottom” and “front” (below). We will now create the isometric and section views as well.



Note each view has a dashed line around it. Selecting and dragging these will move the views if required.

Creating the section view;
Double click the dashed line surrounding the “right view”
Select *Offset section view* icon.
Select a point (at the arrowhead A, above) then double click the second point A.
Moving the mouse will allow you to locate the view as required, click when position is OK.

Creating the isometric view;

Select *window* menu – *tile horizontally*

Orient the part window as required, then select the drafting window to get the drafting workbench.

Select *isometric view* icon and select the piston in the 3D window.

A temporary image is created of the view. You can change the orientation using the compass displayed or select the dashed line and drag the view to a new location. When satisfied left click to accept and the view is created.

Dimensioning the view;

It is possible to make CATIA create dimensions automatically from the sketcher constraints, however many users prefer to generate them manually, giving more control of the drawing. The following describes the manual method of dimensioning.

Maximise the drawing window and double click *dimensions* icon.

Select the lower edge of the piston in “right view”, select and drag to locate the dimension. This dimension is automatically created as a diameter.

Select the inner circle of the hole and select and drag to locate the dimension.

Modifying the piston;

Select *window* menu and *Part1* (or *Piston.CatPart*, if you read the part file)

Double click *Hole1* in the tree.

Change the diameter to 8mm - OK - the piston is updated (or select *update* icon if manual update is optioned).

Double click *Sketch.1* in the tree

Double click the piston diameter D48

Modify this to be 44mm - OK - Exit the sketcher. The piston is updated (or select *update* icon if manual update is optioned).

Updating the drawing;

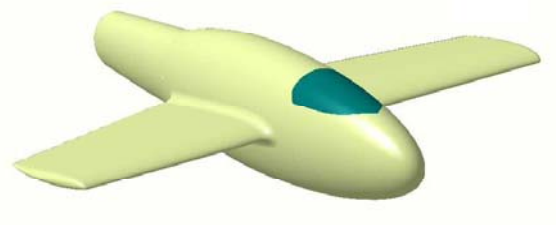
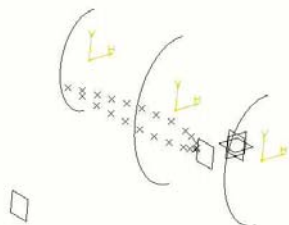
Select *window* menu and *drawing1*

Select *update* icon - the drawing is updated.

7. Using Generative Shape Design – Complex Surface Creation

The following shows how you can create shapes in 3D provided you have good definition curves. It is assumed you have already familiarised yourself with CatiaV5 by following the advice in the previous sections. When creating general 3D shapes such as aircraft, it is essential to spend some time making sure that the curves you use are of good quality and intersect at the right places. For instance, wings should have vertical tangency conditions at leading and trailing edges. A powerful feature of Generative Shape Design (as with all of CatiaV5) is that the whole history of the design is recorded in a specification tree which has modifiable components. For instance, when the design is finished you can swap one of the definition curves for another one and regenerate the design, provided the new curve obeys the same restrictions as the original one (intersection with other curves etc).

Given the curves and points shown we will create the aeroplane shape.



Open the file *C:\CatiaV5_training\UNSW\Aerotute_start.CATIAPart*.

(*Aerotute_done.CATIAPart* is the finished tutorial)

This contains a few curves and points to start you off. It is most important to note that all the green curves are normal to the YZ plane at their ends (they are actually ellipses, created using the sketcher). Notice the contents of the tree. It contains the usual default axis planes, an empty part body, and three open bodies called curves, points and planes_etc. Open bodies contain geometry which is not solid – part bodies contain solids. It is a good idea to create open bodies like these to organise the geometry into manageable groups. The current body being used is underlined, all new geometry will be put there.

7.1 Creating the wing curves

Select *Generative Shape Design* workbench under *Shape*

Right click the curves open body and drag to *Define in work object*. (It becomes underlined, ie current)

Select *work on support* icon (bottom menu) and select *Plane_root* in the specification tree under *Planes_etc* - OK (You are now working in 2D on this plane. The points on this plane for the wing shape have already been entered by selecting *point* icon and using *coordinates* option.)

Select *spline* icon and select each point defining the upper wing shape. (Make sure each point highlights before selecting, otherwise you will input the cursor position)

When you have reached the last point, select the leading edge point in the spline definition window, select *add parameters*, right click the tangent direction entry and drag to Z axis (if necessary select *reverse tangent* for the right direction).

OK to accept the spline.

At any time you can double click any of the spline points and move it or replace it to improve the spline. Use *Update* icon to see the effect of the change. You can also double click the spline and modify its definition.

Repeat the above process for the lower wing points.

We will trim the trailing edge using a corner, so select the *corner* icon and select the top and bottom curve using no trim and a radius of 1mm - OK

Select the *split* icon then the upper curve followed by the corner.

Rename this result as *Upper_wing_root* by right clicking the curve and selecting *properties* - renaming is under *Feature Properties*.

Repeat this for the lower wing curve renaming it to *Lower_wing_root*.

There are many possible ways of creating the wing tip curves. In this example we will project and scale the root curves.

Select *working on supports activity* (under *working on supports*) to return to 3D.

Select *projection* icon with *along direction* option

Select *Upper_wing_root*, *Plane_tip* and *Plane_root* (remember that you can select items like these either in the specification tree or the graphic area, whichever is easier)

OK creates the projection.

Repeat this for *Lower_wing_root*.

Select *scaling* icon, then the top projected curve, the leading edge point of the projected curve as reference and fill in a scale of 0.7 - OK creates the curve.

Rename this to *Upper_wing_tip*

Repeat this for the lower projected curve, renaming to *Lower_wing_tip*.

Later on we will need the trailing edge of the lower wing joined with a line so select the *line* icon and using point to point option select the trailing edge of *lower_wing_root* and *lower_wing_tip* OK

Hide the projected curves by right clicking and dragging to *Hide/Show*.

7.2 Creating the edge fuselage curves

We need to create points defining the nose shape.

Right click the *Points* open body and drag to *Define in work object*

Select *point* icon (double click) with *coordinates* option and fill in 0,-130,0, then OK

Still in the point menu, select the end nose point as origin (just created).

Fill in 0,10,20 - OK

Repeat for 0,10,-20 - OK

Cancel the point menu.

Right click the *Curves* open body and drag to *Define in work object*.

Select *spline* icon and create two more splines (one top, the other bottom) using the ends of the green curves and new points just created, with tangents in the Z axis direction on the nose (reverse the tangent if necessary to get a smooth spline in each case).

We also need a straight line which will be used as a spine for the fuselage surface (not spline) – to create this, select the line icon, select the ZX plane (option *normal to surface* automatically chosen) and select the point on the end of the fuselage nose. Select the infinite line option - OK

7.3 Creating the fuselage surface

Now we will create the fuselage surface as follows;

Make sure no geometry is highlighted (click in open space to deselect)

Select *Insert/Open Body* from top menu - OK

(Notice you can change the “father” for this body if you want to create a more complex tree structure. Also if you multi-select geometry before selecting *Insert/Body* the selected geometry is moved to this new part of the tree. This has no effect on the geometrical result but can be useful for reorganising the tree to a more manageable format)

Right click the new open body and rename it to *Surfaces*.

Select *multi-sections surface* icon

Select each green curve as section curves starting with the smallest and ending with the one nearest the nose points.

In the multi-sections surface window, click in the *guide curve* area (with *guides* tab on) and select one of the two fuselage splines you created, then select the YZ plane to define the tangency.

Repeat this for the other spline.

Select the *spine* tab in the multi-sections surface window and select the infinite line created before.

Select the *relimitation* tab in the multi-sections surface window and deselect *loft relimited on end section*. (This will allow the surface to continue beyond the last section to the end point.)

Select OK and the surface is created.

Right click this and rename it to *front_fuse*.

7.4 Creating the wing

Select the *multi-sections surface* icon.

Select the two top wing curves then OK to create another surface – rename this to *wing_top*.

Select the two bottom wing curves then click in the guide curve area of the multi-sections surface window.

Select the leading edge of *wing_top* and then the surface itself as support. This forces tangency at the leading edge.

Select the trailing edge line drawn previously for this lower surface

OK to create the bottom wing surface – rename this to *wing_bottom*.

Hide the trailing edge line of *wing_bottom*.

Select *Blend* icon

Select the trailing edge of *wing_top* then the surface itself as support.

Select the trailing edge of *wing_bottom* and the surface itself as support.

OK to create the trailing edge surface rename this to *trailing_edge*.

Creating the end of the wing;

Hide the top and bottom wing tip curves

Select *blend* icon

Select the wing tip surface edge on the end of *wing_top* (as first curve) then *wing_top* surface (as first support)

Select the wing tip surface edge on the end of *wing_bottom* (as second curve) then *wing_bottom* surface (as second support). - OK

The small trailing edge tip corner also needs to be created;

Select *blend* icon

Select each of the two edges of the corner together with their respective support surfaces, to maintain tangency.

OK to create the corner surface.

Select the *join* icon.

Select the two wing surfaces, the trailing edge, the wing end and the small wing corner surface - OK. These are now joined and can be used as a single entity (probably called *join1*.)

Rename this to *Wing*

7.5 Filletting the wing to the fuselage

Select *shape fillet* icon

Select the *Wing*

Select the fuselage, ensure the arrows are pointing outwards and fill in 10 for the radius with *trim support1 & 2* options - OK. The surfaces are filleted and trimmed. The trimmed result is a single entity (probably called *fillet.1*) encompassing the fuselage, fillet and joined wing. The original entities are hidden but still listed in the tree.

7.6 Adding the tail area of the aeroplane

Swap to *Curves* open body.

Select *point* icon with *between* option.

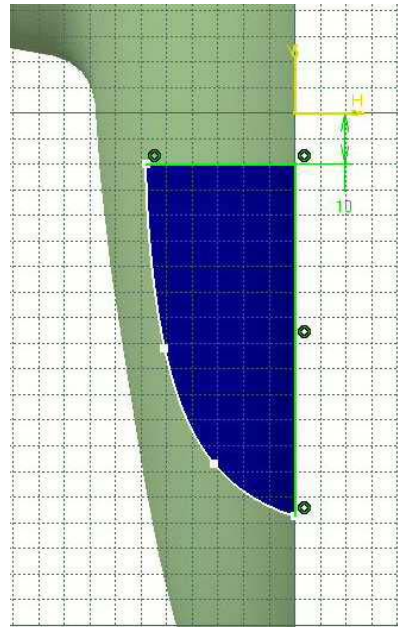
Select the top and bottom ends on the open end of the fuselage.
 Select *midpoint* option - OK
 Select *scaling* icon (double click)
 Select the green curve on the open end of the fuselage and then the midpoint just created.
 Fill in 0.8 as the scale - OK
 Repeat this for the same curve with a scale of .65
 Cancel the scaling menu.

Select translate icon (double click)
 Select the new curve just created at 0.8 scale
 Select ZX plane and fill in 50mm - OK
 Repeat this for the other new curve using 100mm translation.
 Cancel the translation menu.

Swap back to *surfaces* open body
 Hide the rear fuselage (green) curve
 Select *multi-sections surface* icon and select the rear edge of the fuselage and the surface as support, then select the last two translation curves - OK

7.7 Creating a window for the pilot

Swap to curves open body
 Select *sketcher* icon
 Select the XY plane
 Select *spline* icon.
 Using the nose shape as a guide sketch a curve following the nose profile by clicking (say) 4 points as shown, the last point should lie on the centre line (double click this to end the spline.) You can modify this spline later to improve it after the following lines and constraints have been added.
 Select *line* icon and draw two lines from the ends of the spline, one along the centreline and the other perpendicular to it to complete the window profile.
 Select *trim* icon and trim the lines to each other and the spline, making a closed contour. This is essential for the following split operation to work.
 Select *constraint* icon
 Select the vertical line of the profile then the vertical axis then right click and select *coincident*.
 Select *constraint* again then the horizontal line in the profile and the horizontal axis and click again to locate the dimension. Double click this dimension and key in your required dimension (eg 10).
 You can now modify the spline if necessary by selecting any of the points which are not constrained and dragging them to new locations.
 If you wish, you could select *fillet* icon and put a small fillet in the top left of the sketch contour by selecting the spline and the top line.
 Select exit from the sketcher when satisfied.



We will now project this onto the fuselage as follows;
 (A bit of explanation first; operations such as projections sometimes have more than one result. If this is the case you can ask for the nearest solution. However you may not want the nearest solution or there may be two symmetrical solutions. In this case you cancel the nearest solution option and select any geometry which is nearest the solution you want. The result is called near.1 which can be renamed. This is what we need to do for the following projection because the result is two symmetrical curves on the top and bottom of the fuselage.)
 Select *projection* icon, in the projection window the sketch should already be selected as projected element (select if necessary).
 In the projection window select *along direction*.
 Select the fuselage as the *support* (now probably called fillet.1)
 Select the XY plane as *direction*
 Deselect *nearest solution*
 Select OK
 You get a message "2 sub elements, keep only one?" - Answer YES
 Select the point on the top of the nose - OK

You get the window profile on the fuselage, projected along the normal to the XY plane at the location nearest the top nose point (called near1). You could now double click the sketch to modify the shape by dragging the spline points if the projection is not quite right.

Split the fuselage as follows;

Swap to *surfaces* open body

Select *split* icon

Select the fuselage (probably called *fillet.1*)

Select the projection profile and select *keep both sides* in the split window. - OK

You get two results probably Split.3 and Split.4 which are the fuselage with a hole and the window itself.

You can colour the split portion making the pilot's window different by right clicking the surface and choosing properties then selecting graphic to change the colour and transparency - OK

7.8 Creating the other symmetrical half

You can complete the shape by reflecting the surfaces about the YZ plane as follows;

Select *symmetry* icon (double click).

Select one surface and then the YZ plane as symmetry - OK

Repeat this for the other surfaces as required then cancel the symmetry menu.

7.9 Closing the end

Select *blend* icon.

Select one curve on the open end of the fuselage as *first curve*

Select its adjacent surface as *first support*.

Select the other end curve as *second curve*

Select its adjacent surface as *second support*.

Use tangency for continuity on both edges - OK - a perfect blend is produced.

You can clean up your part by hiding all the lines, curves and points. Hint, hide the *curves* open body and the *points* open body by right clicking the tree entry and dragging to Hide/Show.

7.10 Joining everything together

Select the *join* icon and select all the aeroplane parts, rename this as *aeroplane_body*.

Repeat this for the window, naming it *window*.

7.11 Modifying a definition curve

Suppose we want to make the fuselage body wider. To do this we will modify sketch.4. This step demonstrates the real value of parametric modeling.

Double click sketch.4 in the tree.

Select the centre of the ellipse curve in the sketch window (make sure you select it in line with the horizontal axis) and drag to the left. It bulges but maintains the symmetry and end point locations, due to the constraints existing. These are shown as small circles. Bulge the ellipse by the required amount and exit the sketcher.

Select *update* icon to show the result.

You can do this to any of the green ellipses and in addition, because they are ellipses, you can drag the ellipse top point to shrink or grow the ellipse size.

You could make similar modifications to other geometry, for instance double click *Plane_tip* and change its definition to -300 mm offset. Update gives longer wings.

Once you create the part once you can then modify it easily to incorporate design improvements easily

Save your work in your own directory.

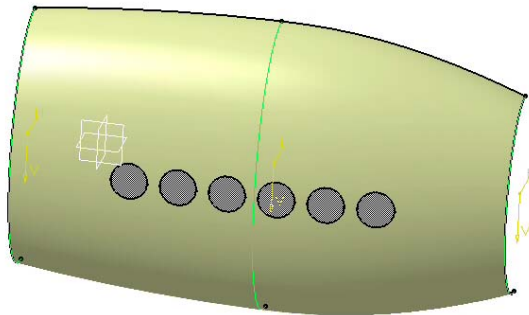
8. Creating Multiple Holes in a Surface

You may want to create multiple holes in a surface. If you were to use the method shown for the cockpit window repeated again and again, it would be very tedious and create a very large file. This is because a copy of the surface is saved for every operation. What you need to do is make the multiple hole process a single operation. The following illustrates a procedure – there may be other methods you could use as well.

Open the part *Aerotute_start.CATPart*

Open *Generative shape design* workbench

Hide the points and create a multi-sections surface using the 3 green curves.
We will now create the holes shown below



Open Sketcher on the YZ plane and sketch the left hole in the position required relative to the multi-sections surface. Make this 15mm diameter.

Still in the sketcher, select *translate* icon (underneath *symmetry*)

Select the circle then select a point on the right of the circle.

In the *Translation definition* window fill in 5 instances, 20mm distance and check *duplicate mode* and *keep internal constraints*.

OK

Now move the mouse to create the orientation required and click to accept.

Exit the sketcher and select project icon

Choose along a direction, make sure *nearest solution* is not checked

Select the multi-sections surface then select the YZ plane for direction – OK

You get a question *do you want only one of the elements*, answer NO (you want them all)

Hide the sketch.

Select split icon.

Select the multi-sections surface

Select the projection and check *keep both sides* – OK

You now have two split elements, one is the multi-sections surface full of holes and the other is all the holes as one entity.

You can change the colour and transparency of the holes by right clicking and selecting properties/graphic.

This method could also be used for a sketch containing holes of different size, shape and spacing.
This will also work on a set of joined surfaces.

9. CNC Milling (3 axis)

The School has three CNC milling machines, a Makino 3 axis, a Fadal 3 axis and a DMG 5 axis. When creating any tool path you must take into account which machine you will be using – there may be slight differences in the final program. The most obvious difference is the X, Y and Z travel.

The Makino can move 800mm in X, 400mm in Y and 400mm in Z.

The Fadal can move 1000mm in X, 500mm in Y and 700mm in Z.

The DMG can move 600 mm in X, 600 mm in Y, 600 mm in Z, +/-360° in C (rotation of tool about Z axis) and 90° in B (rotation of tool about Y axis)

All CNC programs must be checked by Laboratory staff before running them on the machines.

You should ascertain the best available tooling and feeds and speeds to be used – see Laboratory staff for the latest table of available tools and recommended machining conditions.

The techniques below are for 3 axis machining and generally apply to all three machines. There are some additional techniques for the 5 axis machine which you should check out in section 12 when you are competent in 3 axis programming.

9.1 Simple Machining (3 axis)

CatiaV5 has prismatic and surface machining modules for CNC milling machines. The following is a simple example using prismatic machining.

Open C:\CatiaV5_tutorial\UNSW\Simple_machining.CATIAPart

This part contains a simple solid with a pocket and an island.

Looking at the specification tree, the solid is called *final_product* and there is another solid called *work_material* (in hidden mode) which has been created for the machining simulation.

The part also contains an axis created at the top/centre of the *work_material* – to be used as a machining axis, and a plane to be used as a safety plane.

Before you start machining, check some of the CATIA options.

All 3 axis toolpath output from Catia must be **tool tip**. This is probably the default setting but check it anyway to make sure – select *tools/options/NC_manufacturing/output* and check *tool tip* for *tool output point*. If you do not save your Catia settings, this and all other options will have to be checked every time you start Catia (unless you have your own computer running Catia).

Select *Tools/Options/NC Manufacturing/Output*

Then select *Cenit* for the Post Processor and *Store tool path in current document* - OK

You should also fill in the path name for all toolpath files which must reside on your Z drive if you are not using your own computer.

Note if you intend processing very large NC tool paths you should use *Store tool path in external file*

Select *Prismatic Machining* workbench. (Note there are 3 possible workbenches; Prismatic, Surface and Advanced – Advanced combines Prismatic and Surface in one workbench, hence you could use *Advanced* for everything) After selecting the workbench a *CatProcess* type file has been opened automatically which should be saved separately when the toolpath is complete. The tree contains an empty *Part Operation* and *Manufacturing program*.

Double click the *Part Operation* and in the *Part Operation* window select *machine* icon.

In the *Machine Editor* window ensure that *3_axis mill* is selected and

Select the required Post Processor and Post Processing table ie FADAL or MAKINO.

The rest of the settings can be left as default.

Note in the *Spindle* tab the spindle orientation for both machines is 0,0,1

Note also that all circle options for circular interpolation are selected except for 3D.

OK to accept.

Now select the *Reference machining axis system* icon.

Select the red shaded axis and then the axis system required ie the one created in the centre on the top of the *work_material* - OK. (Note this axis is located exactly on the top of the raw material block. This is a recommended strategy but it is possible slight variations of this will be preferred by the machine operator, depending on the machining to be done. Check with Laboratory staff for the preferred axis location).

Select the *stock* icon then *Work_material* in the tree (under *Product List*), double click to end selection.

Select *safety plane* and select the plane above the job..

OK (There are other settings possible which can be investigated at a later time)

You now need to insert a post processor command to create a fixture origin statement. This will allow the machine operator to set the job anywhere which is convenient on the machine work table. The machine will then be set to have a fixture origin which is the same as your machining axis origin. The fixture offset can have any number from 1 to 5. With the *Manufacturing Program* highlighted use *add post processor instruction* icon and write ORIGIN/MANUAL,0 (for offset 1) - or

ORIGIN/MANUAL,1 (for offset 2) etc.

You must inform the machine operator which offset number you are using as well as the location of your machining axis system.

It is also most important that the tool geometry used in Catia is exactly the same as the actual tool on the machine and that the machining conditions are correct – obtain a copy of the latest tool table from workshop staff to ensure this. An example of a tool table is also shown in the Appendix at the end of this document.

Select *pocketing* icon – we are going to machine the pocket with island.

There are 5 tabs along the top of the pocketing window, we will select each one in turn starting from the left, but you can in fact select them in any order or return to them later to modify a parameter.

Tab 1 – machining strategy

Select the *strategy* tab (top left tab in the pocketing window) and fill in the required machining options. For this exercise we will just set the options in the *axial* tab to *maximum depth of cut* 3mm (do not select OK yet, if you do by accident, double click the pocketing entry in the tree to open the dialog again). Like all CatiaV5 entries you can revisit any of these areas and modify them later if necessary.

Tab2 – workpiece definition

Select the next tab on the left at the top of the *pocketing* window.

Select the bottom of the pocket image then the bottom of the actual pocket.

Select the top rim of the image then the actual top surface of the pocket.

(The island will probably have been detected automatically, but if it hasn't select the image of the island then the actual island in the part. You can tell which parameters still need defining if they are still red and not green)

Tab3 – tool definition

You can modify the tool to be used with the next tab changing its dimensions, name and number (the tool number must correspond to the number used by the CNC machine operator).

Tab4 – machining conditions

The machining conditions can be modified with the next tab. You should check the best conditions to use (consult the tool table in the Appendix at the end of this document and the latest version available from Laboratory staff) – uncheck *automatic compute* to enter your own values.

Tab5 – macro definitions

Select the last tab on the right for macros – these control the tool's motion when not actually machining. It is extremely important to get macros right – most machine crashes occur from bad or absent macro definitions. The pocket operation is plunging the tool into the work material. This is not recommended for most milling tools (with the exception of slot drills). Suppose we are not using a slot drill which can plunge vertically, so we must change the way the tool approaches. To do this, select the approach macro in the scroll bar, then the predefined helical macro (looks like a zigzag motion).

Change the ramp rate to 10 degrees – the tool will now spiral down at 10 degrees instead of plunging in vertically.

Define the retract macro as a vertical movement of say 50 mm using the predefined vertical motion macro.

There is one more macro to define which controls the motion during the pocket operation. In this pocket the operation might require the tool to jump over the island. We must specify how this is to be done – if we do not do this it might travel through the island or plunge in a way we do not want.

Select *return in a level* macro. This has two parts – a retract and an approach. Scroll to *retract* and make it vertical 50mm. Then scroll to *approach* and select the tool kit tab labelled *build a user macro*.

This displays icons for making macros of your own design – place the cursor over each icon to display what they do.

You can put multiple operations in the macro by successively selecting icons and choosing their parameters. A macro is always built starting at the workpiece and moving away, regardless which type it is. Choose the *add ramping motion* icon and make it 10 degrees as with the main approach macro.

Calculating the tool path and replaying

Select *Replay* icon in the pocketing window – the tool path is calculated and stored.

You can now replay a wire frame rendition using the replay buttons in the replay window or select the *video camera* icon and replay a solid version (the solid replay uses the *work_material* specified in the *Part Operation*).

Select OK to exit pocketing.

You could double click *pocketing* in the tree and look at other options and experiment by changing them and recomputing the tool path.

Further possible machining on this component

So far this process has only machined the pocket – the top face of the edges of the pocket and the top of the island have not been machined. You might prefer to machine the whole of the top first which can be done by inserting a *facing* operation before the *pocket* operation and selecting the top face. If you prefer to machine just the remaining top edge face then place a *sweeping* operation after the pocket operation. (*Sweeping* is accessed using the *surface machining* workbench or you can select *advanced machining* workbench which contains all possible machining methods in one workbench)

To machine the top of the island you could also use sweeping. The island top is 5 mm below the top face so choose an axial strategy which makes 5 cuts at 1mm deep. Choose macros which will make the tool come in from outside to avoid plunging onto the island.

See section 10 for generating NC code from a tool path.

9.2 Surface Machining (3 axis)

If you intend to mill a very large part that will generate a tool path with a very large number of points, before starting you should go to *Tools > Options > General* and:

- in the *General* options page, deactivate *Automatic save* and *Load referenced documents*,
- then go to the *Performances* tab and set the *Undo stack size* to 0 or 1,
- finally go to *NC Manufacturing > Output*, activate *Store tool path in an external file* and deactivate *Store contact points in tool path*.

You should return the *Undo Stack* to 10 when you have finished your large machining tasks.

Open file *C:\CatiaV5_training\UNSW\Aerotute_done.CATIAPart*

We will assume you wish to machine one of the wings which will later be assembled on the fuselage as a wind tunnel model. To do this you must create some more features to use.

First you must decide how the wing will be attached and therefore where and how it will end. This example will machine the wing and wing root 28mm from the fuselage centreline. This is not necessarily the best mechanical way of fixing the wing but illustrates techniques to be used. Also when machining a wing like this you must be able to machine both sides and hold it rigidly. The best way of doing this is to use a raw material block which is longer and machine so that the wing remains attached at both ends. Hence in this case the blend on the wing tip will have to be hand finished and the root portion cut off with a band saw. To allow extra material to hand finish the wing tip, you need to extend a portion of wing tip as shown in the next section. (You could cut the wing tip off square and machine a wing tip component separately, fixing it with dowel pins and locktite. Each machining problem is usually somewhat unique, so use your ingenuity!)

9.2.1 Preparing the model for machining

The following steps will create more geometry. You might like to create a new open body called *NC_geometry* to contain this geometry. A tidy product is much easier to manage and others can understand it more easily.

Extending the wing tip.

Select *Generative Shape Design* workbench and select *extrapolate* icon.

Select the wing tip edge on the top, then select *wing_top* surface (make sure you select the actual multi-sections surface which is currently hidden but accessible via the tree)

Select *length* option and 20mm - OK

Repeat this for *wing_bottom*

Select *extrapolate* icon

Select the tip edge of the trailing edge, then select *trailing edge* in the tree (it is hidden).

Fill in 20 mm – OK – gives wing tip extension to be included in the machining (top bottom and trailing edge surfaces).

The machining will include a portion of the fuselage to a plane 28mm from the centreline of the fuselage. A raw material block will be created to suite the resulting machining operation. Note it is always best to choose a raw material block which matches the shape required as closely as possible to reduce the amount of machining which is not necessary. You do not want to spend valuable machining time machining large volumes of material into chips unnecessarily.

With the *Generative Shape Design* workbench create a plane -28mm offset from the YZ plane. Intersect this with the *aeroplane_body*

Select *Part design* workbench and open the sketcher on the XY plane.

Sketch a trapezium surrounding the wing and extension. Constrain the edges to be coincident with the extended wing tip and the fuselage intersect curve. (Hint, use the sketcher *constraint* icon, select the two lines to be coincident then right click and drag to *coincident*, for the tip, select the sketcher line and the trailing edge tip point of the extended wing).

Exit the sketcher and create a pad to surround the wing eg 25mm up and 15mm down.

Select sketcher on the fuselage intersect plane and create a rectangle surrounding the intersect curve.

Exit the sketcher and create pad to reach the wing root from the intersect plane.

These two pads will be the stock material.

Select *point* icon in the *Generative Shape Design* workbench and select two opposite corners of the top of pad.1.

Using *between* option create a point halfway between these corners.

Select *Insert/axis system* (top menu) and select the midpoint just created – this will be our machining axis.

Repeat this for the under side of the workpiece (reversing the Z axis and keeping the X axis the same) since we will be turning this over to machine the other side. Note it is important to know the distance between these axes (ie the stock thickness) because this must be the same on the actual raw material.

The finishing cut will require a closed profile to control the limits of machining. Open the sketcher on the XY plane and sketch a rectangle surrounding the machining zone. Constrain the root end to be coincident with the fuselage intersect curve – rename this *finish_profile*.

Finally you need a plane created which will be used as a safety plane (the plane at which the tools cannot collide with anything). So create a plane using the *plane* icon say offset 100mm from the top of the block (where the axis is).

Repeat this for the under side of the block.

Finally create a solid rectangular block which represents the actual work material to be used and will be used in the simulation. Using the *Part Design* workbench create two pads in a new part body 25mm longer at the wing root and tip. These 25mm extra ends will be used to hold the work material when machining.

Rename this part to *simulation_workpiece* and hide the geometry.

(Note we have created one part body which is used to calculate the actual tool path and another part body which represents the actual work material which will be placed on the machine)

Save the Part as filename *Aerotute_done_nc* in your own Z drive. You can read *Aerotute_done_nc* from *Catia_training\UNSW* if you have problems with this then save it in your own Z drive

9.2.2 Machining the top wing surface

Surface roughing

Before you start machining, check two of the CATIA options (if you save your settings, this need only be done once).

Select *Tools/Options/NC Machining/Output*

Then select *Cenit* for the Post Processor.

For reasons which will become apparent later we will create the tool path from a Product. A Product contains assemblies of entities, such as Parts.

Select *File/New* then *Product*

Rename this to *Aerotute_nc*

Select *Insert/Existing Component* and select *Aerotute_done_nc*

Select *Surface Machining* workbench (or *advanced machining* workbench).

A *CatProcess* type file has been opened which should be saved separately when the toolpath is complete (name it *Aerotute_nc* same as all the other related files). The tree contains a *Product List* and an empty *Manufacturing program*.

Rename *Part Operation.1* to *Top_wing_machining*.

Double click *Top_wing_machining*.

In the *Part Operation* window select *machine* icon.

Ensure that *3_axis mill* is selected.

Select the required Post Processor Table and Post Processor ie FADAL or MAKINO.

The spindle orientation for both machines is 0,0,1

OK to accept.

Now select the *machining axis* icon.

Select the red shaded axis and then the axis system required ie the one created in the centre on the top of the stock block - OK

Select the *stock* icon then *Simulation_workpiece* in the tree, double click to end selection.

Select *safety plane* and select the plane at 100mm above the machining axis created before.

OK

You now need to insert a post processor command to create a fixture origin statement. This will allow the machine operator to set the job anywhere which is convenient on the machine work table. The machine will then be set to have a fixture origin which is the same as your machining axis origin. The fixture offset can have any number from 1 to 5.

With the *Manufacturing Program* highlighted use *add post processor instruction* and write *ORIGIN/MANUAL,0* (for offset 1) - or

ORIGIN/MANUAL,1 (for offset 2) etc.

You must inform the machine operator which offset number you are using as well as the location of your machining axis system.

Select *Roughing* icon.

Roughing.1 is added to the tree and a definition menu appears.

Select the *strategy* tab (top left tab in the roughing window) and fill in the required machining options. Select the ? in the menus for help on the meaning of the various parameters. For this exercise use course values eg 1mm machining tolerance and a large step over eg 8mm with 10mm depth (we will be using a 10mm diameter tool). This will speed up the simulation. When you are happy with the tool path you can refine the tolerances – this method speeds up the optimisation of your tool path by making the initial calculations and replays much faster. Like all CatiaV5 entries you can revisit any of these areas and modify them later if necessary.

Select the next tab on the left at the top of the roughing window.

Select *Part* and select (from the tree is easier) all the surfaces to be machined ie *Aeroplane_body*, *extrapolate.1* and *extrapolate.2* - OK.

Select *Rough stock* and select *PartBody* in the tree.

Change the *offset on part* to 3mm (there is no check surface used in this example, a check surface is used if you wish to avoid something when machining).

Select *safety plane* and select the 100mm plane above the job.

Select *bottom* and select a bottom corner of the pad surrounding the wing.

Select the centre top tab (tool definition) and (if necessary) change the tool to 10mm diameter end mill (start by typing in a new name, then you can change the dimensions). This window also has tool number which can be changed. Generally it is best to use number 1 as tool 1 and 2 as tool 2 etc.

Make sure you inform the machine operator which tool numbers you used and consult the tool table (see Appendix) for available tools and recommended machining conditions. If the tool you are specifying has a small diameter it will be much shorter than larger diameter tools and therefore may not be able to reach all the locations you are specifying.

Select the next top tab on the right and fill in the required machining parameters (uncheck *automatic compute* to enable the parameters to be modified).

The last tab is for machining macros which control the tool's movement when not actually machining ie approach, retract etc. Do not plunge vertically into the work unless you are using a slot drill or a ball end mill with a small plunge value.

Select *Replay* in the roughing window – the tool path is calculated and stored.

You can replay a wire frame rendition using the replay buttons or select the *video camera* icon and replay a solid version.

Select OK to exit roughing.

Surface finishing

Select *sweeping* icon to do the finish cut

If necessary select the first tab on the left at the top of the sweeping window.

You have three other tabs in this window, *machining*, *stepover* and *machined zone*.

Fill in the required values, using the ? for help.

Use 1mm scallop height, 1mm minimum distance to speed up the simulation. When happy with the tool path you should refine these to smaller values for your actual machining program eg .01mm scallop height and .01mm machining tolerance.

Select the next tab on the left at the top of the sweeping window

Select *limiting contour* and select the sketch *finish_profile* (in the tree is easier) - OK.

Select *Part* and select all the surfaces to be machined again - OK.

Select *Top* and *Bottom* points as before by selecting the tip and bottom corner points of *PartBody*, the raw material block

Set *offset on part* and *offset on check surface* to be 0.0

Notice you can control how the tool behaves at the limiting contour. You can machine *inside*, *on* or *outside* the contour, and the stop mode can be contact point or tool tip and you can also specify an offset to be added.

The next tab allows you to change the tool (key in Ball Mill and select ball mill option).

The next tab is machining feeds and speeds.

The final tab is used for macros, ie how the tool will move when not machining.

Set all the other tabs as required, making sure your macros give safe tool paths for moving around and to the job.

Select *Replay* to calculate the tool path then replay the video of the whole tool path.

When the video is finished, select the disk icon and save the replay in file *Top.cgr*. This is a representation of the solid resulting from the machining process which we can use as a raw material for the next operation.

9.2.3 Machining the bottom wing surface

Surface roughing and finishing

Using the *window* menu at the top switch to the Product window.

Highlight *Aerotute_nc* in the tree then

Select *Insert/Existing Component*.

Select *Top.cgr* - this inserts our new simulation work material created from the previous operation.

Switch back to the NC setup editor using the *window* menu.

Rotate the model upside down with the mouse.

Highlight *Top_wing_machining* in the *Process List* and select *Part Operation* icon.

A new part operation is created in the tree and the geometry vanishes because no geometry has been assigned to this new operation yet. Rename the new operation to *Bottom_wing_machining*
Double click this and select 3 axis machine, ptable and post processor as before and then select the icon opposite *No Product or Part selected*.

This opens the file selector – select *Aerotute_nc* Product (the geometry reappears)

Change the machining axis.

For stock material select *Top.cgr* (this will enable the next verification to use the pre-machined workpiece)

Fill in all the other details as before.

Ensure that *Bottom_wing_machining* is highlighted and select *manufacturing program* icon to insert a new machining program.

Repeat all the steps for roughing and finishing as before but using the new axis system, new safety plane etc (don't forget the first ORIGIN statement). **Note**, in the strategy tab (top left in the roughing or sweeping window) you will need to select the vertical arrow on the tool image and select *reverse axis*. This will turn the tool upside down.

Selecting the new axis system at the start does not change the tool orientation. This will need to be done for the roughing and the sweeping operations.

Replay the tool path in video mode to see the final product. You could save this as before as a .cgr file for future reference – perhaps to show the NC machine operator. Obviously, you can improve the quality of the result by reducing the scallop height and machining tolerance in both the sweeping operations. Try machining tolerance 0.01mm and scallop height and minimum distance 0.1mm.

Note, the roughing in this example on the top operation carries through to the bottom. Hence when roughing the bottom there are redundant roughing paths (material has already been machined). One way to avoid this would be to create a check surface passing through the corner points of the wing. Then select this surface as a check surface in both the top and bottom roughing operations.

If you have problems with any of this you can read the files;

C:\CatiaV5_training\UNSW\Aerotute_nc.CATIAPProduct

C:\CatiaV5_training\UNSW\Aerotute_nc.CATIAPProcess and check them out.

Note, you can break up your operations into individual *Manufacturing Programs* by inserting other manufacturing programs using the icon. This is useful for large machining jobs because the regeneration of NC code after modifications is limited to a smaller section of the tool path – you don't have to regenerate everything every time you make one small change. However, since your tool path is now a number of separate files, to check the replay properly you will need to generate aptsources and check it with Vericut (see sections 9 and 10), which accepts any number of input files. You should rename each *Manufacturing Program* to help keep track of what they are.

10. Generating NC Code (3 axis)

You now need to generate NC code from the previous tool paths.

Note - you must have saved the *CATIAPProcess* and associated *CATIAPart* and (if used) *CATIAPProduct* before generating NC code – the process uses the disk copy of these files when calculating the code – use *File/Save All*.

Use the *CATIAPProcess* you created before or open the file *C:\CatiaV5_training\UNSW\Aerotute_nc.CATIAPProcess*

Highlight the *PartOperation* and select *Generate NC code in batch mode* icon

Select *NCCode* as data type

Select *Output file* and fill in required name and location.

Select *Options* tab and select *Z axis circles* for circular interpolation.

Select *NCCode* tab and select FADAL or MAKINO machine (depending which machine you selected in the machine definition in the Part Operation).

Select execute – the NC code is written to the file specified.

Note, at present the generation of NC code always has an error at the beginning of the program (a line containing “G00 X0 Y0 Z100” - something that CATIA so far cannot fix – delete this line).

Your data (Z drive) can be accessed on the computer next to the machine and hence the program can then be transferred for machining.

11. Verification of Tool Paths using Vericut (3 or 5 axis)

Vericut is an independent tool path checking utility which has advantages over Catia. It is 10 times quicker for large tool paths and allows sophisticated checking for occurrences of bad machining practice which are not apparent in the Catia replay. Vericut must be used as a final check for all programs destined for actual machining.

Start Vericut using the desktop icon – if you get a licence failure message chances are all the 5 available licences are in use. Since there are only these 5 licences please exit Vericut when finished and do not “hog” it so that others can get a chance to use it. Vericut requires the aptsources file generated by Catia (an aptsources file is created when you create the nc code file as described in section 9 above).

Setting up Vericut

Start Vericut

Select *File/new session* to set an empty session

Select *Setup/toolpath* and choose the *aptsources* file required. You can have more than one file, they will be replayed in the order they appear.

In the same toolpath window, select Toolpath type as Catia APT

It is useful to see the axis systems displayed, so select *View/Axes* then *set all*.

Select *Model/model definition* and fill in;

- Active component = *stock*
- In the model tab, type = *block*
- Fill in X Y Z size of your stock block then click *add*
- In position tab select *component* and select a translation which puts the block in the correct position relative to your axis system. For example if your block was 100 x 100 x 50 and your axis was top dead centre then the component translation is -50, -50, -50. (Note, default axis position is always bottom corner of the block)

Select OK

Note that in the model tab above you can select *cone*, *cylinder* or *model file* instead of *block* if a more complex stock is required. Choosing *model file* allows you to use a solid from Catia as stock which has been saved using *File/save as STL*.

Now select *Setup/motion* and set

- *stop at max errors*=1
- set fast feed rate to be greater than your fastest feed chosen – this will be considered a dangerous feed when in contact with your job

Select OK

All this set up can be saved as a single file so you can come back to this at another time by simply reading one file. To do this select *File/save user* and fill in a *file.usr* name with a path that you can recover later (eg your Z drive). Reading this will restore all the settings you now have.

Select *Info/VERICUT.log* which will give you either an empty file or the previous Vericut log – in this window select *File/reset log*.

The replay will now stop if it finds an occurrence of fast feedrate on the workpiece and will display it in red, it will also put the occurrence in the log file for you to examine (*Info/VERICUT.log*) after the replay. *Info/Toolpath* will show you where in the program you are so you can find the offending code.

Hit the play button on the bottom right and control the replay with the buttons there.

12. DMG 5 axis machining

Before tackling this section, you must be competent in 3 axis machining as described in sections 9 and 10.

The DMG 5 axis machine can move in X, Y and Z (the same as the other machines) but with the option of rotating the table +/-360° (C axis, rotating about Z axis), and rotating the spindle axis 90° (B axis, rotating about Y axis). To achieve these rotations you simply specify how the tool should move in the menus and the post processing takes care of translating this motion into X,Y, Z, B and C commands. You should make your Z zero on the top of the work material as with the other 3 axis machines. Use the Advanced Machining workbench. The following guidelines show the additional techniques required.

- The 3 axis post processor command ORIGIN/MANUAL,0 is not required.
- The following post processor multiple command is required;
 - RAPID
 - GOTO/0,0,100,0,0,1
 - MODE/TLVEC,ON,50000

These commands can be combined in one post processor entry in the program and must be used before each operation unless already invoked before in the previous operation. Any tool change will automatically cancel these instructions and so they must be restated after every tool change. (Use copy and paste in the tree)

- Take great care with tool motion macros. The tool's motion when not cutting on the machine is not necessarily the same as the Catia or Vericut replay. To be safe, always move up to a safety plane for retraction and down from a safety plane for approach. This can also apply to the other macros such as *return*

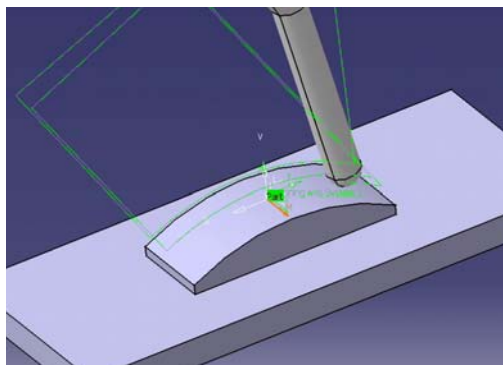
in a level, depending on the situation. Take care when specifying the motions – for instance, when retracting to a safety plane, sometimes axial motion is best and other times motion normal to a plane is best.

- It is most important that the tool geometry used in Catia is exactly the same as the actual tool on the machine and that the machining conditions are set correctly – obtain a copy of the latest tool table from workshop staff to ensure this. An example of a tool table is also shown in the Appendix at the end of this document.
- All toolpath output from Catia must be **tool centre**. This is one of the options in your Catia settings – select *tools/options/NC_manufacturing/output* and check *tool centre for tool output point*. **Remember that this is a setting which will now apply to all CNC programs you create. So if you want to create a 3 axis program for the Fadal you will have to set this option back to tool tip.** If you do not save your Catia settings, all options will have to be checked every time you start Catia.
- You must specify a 5 axis machine in the *Part Operation* menu and select ICAM_MILL_MM for the post processor words table, do not select anything for post processor.
- Although you are using the 5 axis machine, the machining operations can still be 3 axis. You can however set the spindle angle to be different (but fixed) from the Z axis using a 3 axis machining algorithm. Hence for instance you could create a three axis roughing path but at a different fixed tool angle. Variable tool angle 5 axis operations can be accessed by selecting the small black arrow in the corner of the machining menu icon – this reveals the 5 axis optional icon. You then need to specify different parameters as specified in the new machining menus. It is best to use a ball end mill for all variable tool angle 5 axis operations. Other types of tools may result in gouging – discuss your program with Laboratory staff for more help on avoiding gouging. Note that gouging of this type is not always shown in the verification replays (see discussion and figures below).
- Because you have control of the tool angle you should try to avoid machining on the tip of ball end mills, lean the tool over so that the contact point is nearer the tool diameter.
- All tool path output from Catia must be in APT format (*file.aptsource*). Use the same method as described in section 10 but select *APT* instead of *NCCode*. This is converted to CNC code at the computer next to the DMG machine.
- Your data (Z drive) can be accessed on the computer next to the 5 axis machine.

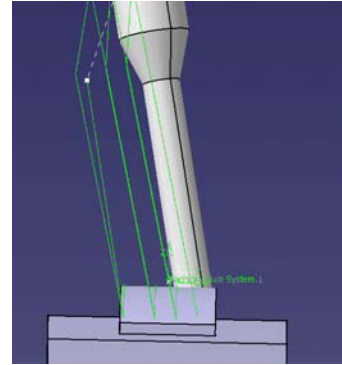
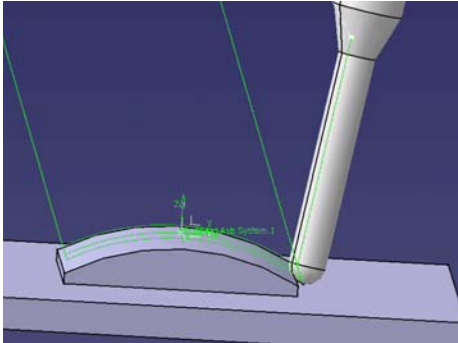
A possible problem with 5 axis variable tool angle programming

You should try to avoid tool paths where the tool angle passes through the vertical during a cutting operation (ie goes from +angle to –angle relative to the Z axis). This can result in rotations of the table which are larger than ideal (typically 180°) while the X, Y or Z values only change by less than 1 mm. This could gouge the workpiece and extends the machining time considerably. (Note this problem obviously does not apply to machining at any fixed tool axis orientation). This large C axis motion is not evident in the Catia or Vericut replay, it can only be seen in the CNC code itself. There is a program available on the PC next to the 5 axis machine which can be used to find occurrences of large C axis variations in the CNC code. Using this you can pin-point problem areas then revisit your Catia program to try to improve the tool path. There are no simple guidelines which will avoid this situation, it depends on the cutting parameters you have chosen and the geometry and orientation of the part.

The figures below show a simple example of this problem. They show a sweeping over a cylindrical surface with a ball cutter, in the first figure the tool is leaning forwards at 31° to the surface normal which means for one pass the tool is always leaning forwards relative to the Z axis (the normal to his surface at the start is 30° to the Z axis). If the tool path had been programmed using zigzag motion instead of one way the tool would have reversed its angle through the Z axis on the second and subsequent passes and therefore produced 180° C axis motion at the end of every pass. On the Catia replay zigzag looks more efficient but on the 5 axis machine one way machining is much faster in a situation like this because the return path is at rapid traverse and the many 180° rotations are slow.



If this lean angle is changed to 20° (or any angle less than 30°) then the tool does pass through the vertical as in the figure below left. This will result in a 180° turn of the table at the transition point on every pass.



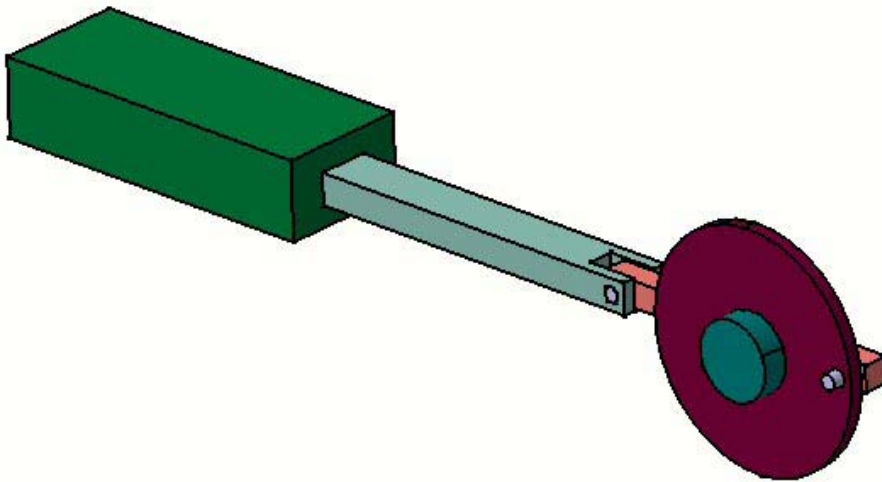
The figure above right shows the same tool parameter of 20° but with an additional tilt sideways of 10° . The extra tilt avoids the tool passing through the Z axis and so avoids the 180° C axis movement. You should note however that this does result in steps of up to 15 degrees in the C axis (with XYZ motion less than 1mm) while machining, so the first option of a lean of 31° is best in this case.

For more complex parts you could try limiting your machining to a series of smaller zones to keep better control of the tool angle variation.

13. Kinematic Mechanism

13.1 Single Degree of Freedom Mechanism

To create a kinematic mechanism as below you need an assembly of parts, hence you need to create a product. If you have trouble with creating this, read *Mechanism_done.CATIAProduct*
Select *File/New* and *product* as the type – OK



Select *DMU Kinematics* workbench

Highlight *Product.1* in the tree and select *Insert/Existing component*

Using the file selector get part *C:\CatiaV5_training\UNSW\mechanism\Body.CATIApart*

Repeat this for *Slider.CATIApart*

Notice that *Slider* has loaded inside *Body*. You can move parts around using the compass (the axis system in the top right of the window). To use the compass, select the red dot at its centre and drag it to a face of *Body*. This should place the compass on the face and it should be green. If *Body* is highlighted, selecting and dragging the active areas of the compass will now move *Body*. So move it to separate the two parts. You can return the compass to its normal position by selecting and dragging the red dot away from *Body*.

Using *Insert/existing component*, load *Rod.CATIApart*, *Wheel.CATIApart*, *Axle.CATIApart* and *Pin.CATIApart*.

Separate them in the window using the compass to move them around. Make sure the part you want to move is highlighted in the tree.

Finally load another copy of *Pin.CATIApart* separating it from the other *Pin*.

You are now ready to assemble and constrain the parts. Note, if in any of the following assembly processes it is difficult to achieve the correct orientation then use the compass to move the component to nearly the right orientation first.

Using the *DMU Kinematics* workbench, select *Insert/New mechanism*

Mechanism.1 appears in the tree under *Applications*

Select *Prismatic Joint* icon (you may have to expand the *joints* icon to find it)

Select a longitudinal edge of *Slider* and the required matching edge of *Body*, then a face of *Slider* and the required matching face of *Body* – OK, the joint is created.

Selecting *Update* icon will assemble the two parts (use the compass to adjust the *Slider* position if required)

Select *Revolute joint* icon

Select a hole in *Rod* (axis is visible) then the hole in *Slider* (axis is visible).

Select a face on *Rod* and the required matching face on *Slider* – OK

Update to see the assembly.

Now using the compass align *Slider* and *Rod* in a straight line.

Select *Revolute joint* icon again

Select the hole near the rim of *Wheel*

Select the hole in the end of *Rod*

Select a face of *Wheel* and a required matching face of *Rod*. – OK

Update to see assembly – if you have problems getting the right orientation of *Wheel*, use the compass to roughly orient it first.

Now place the compass on the axis of the matched holes and in line with the axis., with *Wheel* highlighted, rotate it so that *Rod* passes across the centre of *Wheel* (as in the figure above).

Select *Revolute joint* icon again

Select the hole in the centre of *Wheel* and the small boss on *Axle*.

Select the outer face of *Wheel* and the face containing the small boss on *Axle* – OK

Update to see assembly.

We now wish to lock *Body* and *Axle* with respect to each other, so select *Rigid joint* then select *Body* and *Axle*

The mechanism also needs to have one of its components fixed, so select *Fix* icon then *Body*.

This will be a single degree of freedom mechanism and so needs to have one variable to drive it. You can choose any of the joints as the driving one, the following uses the wheel rotation.

Double click in the tree *Revolute.4(Wheel.1,Axe.1)*.

In the joint definition window check *Angle driven*

You can also change the angle limits if required – OK

You should now get a message *Mechanism can be simulated*.

So far we have left out the pins for the joints on both ends of *Rod*. They are not necessary for the mechanism but you can include them for realism.

To do this select *Assembly* workbench.

Select *Coincident* icon then select the hole axis and pin axis.

Select *Coincident* icon again and select an end face of *Pin* and side face of *Rod*.

Update to see assembly.

If *Pin* is the wrong way round, double click the constraint in the tree, select *more* and modify the orientation.

Repeat this for the other pin and other hole in *Rod*.

Finally the pins must be locked to *Rod* and *Slider* so that they will move when the mechanism moves, so go back to *DMU Kinematics* workbench, select *Rigid joint* icon and select *Rod* and the pin on *Wheel* rim and repeat for the other pin, selecting *Slider* and *Pin*.

You are now ready to simulate your mechanism.

Select *Simulation with commands* icon

If necessary select *more* to get the full menu possibilities

Check *On request* option, move the command slider to a new destination angle and press play.

If you want to drive your mechanism using the *Slider*

You could double click in the tree *Revolute.4(Wheel.1,Axe.1)*.

Uncheck *Angle driven* – OK

Double click in the tree *Prismatic.1(Slider.1,Body.1)*

Check *Length driven* – OK

Simulate as before.

If you need to change dimensions affecting the mechanism or for instance move *Body* a different distance from *Axle*, you must first delete the constraints associated with them. Make the change either parametrically in the normal way or move something with the compass.

Then redefine the constraints you deleted to recreate the mechanism.

There are many more joint types which you should investigate yourself.

13.2 Adding a Further Degree of Freedom

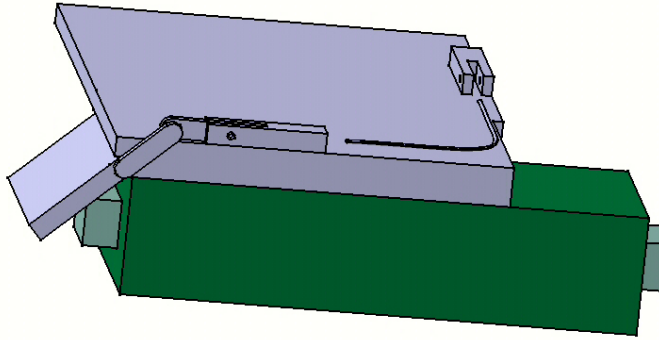
Using the mechanism you created above (or read *Mechanism_done.CATIAProduct*) we will add another degree of freedom.

Select *Insert/Existing Component* (if necessary highlight *Product.1*)

Select *Frame.CATIAPart*

Select *Assembly Design* Workbench

Using *Coincident* menu, select the required face and edges then *Update* to position *Frame* as in the figure.



Select *Insert/Existing Component* (if necessary highlight *Product.1*)

Select *Door.CATIAPart*

Return to *DMU Kinematics* workbench

Using *Revolute Joint* menu and *Update*, create the assembled joint as in the figure – hint, make sure you get axis images by selecting the fillet faces to be matched and also select the side faces to be matched.

Continue with the other parts (*Cable1* and *Lever*, inserting *Cable1* twice). Note *Lever* should have a *Revolute joint* with *Cable1* and a *Cylindrical Joint* with the pin of *Door*. Also you should first create *Prismatic Joints* for *Cable1.1* and *Cable1.2*, these can then be converted to a cable joint using *Cable Joint* icon. Use the compass to change the position of parts as necessary.

Finally create a rigid joint between *Frame* and *Body* and modify the cable joint to be the driving input for the second degree of freedom.

This should give you the mechanism required for the simulation exercise below.

14. Creating a Kinematic Simulation

Having created your mechanism you can create a simulation which can be replayed. This is particularly useful if your mechanism has more than one degree of freedom. Using the simulation you can specify how the multiple inputs will be applied.

To illustrate this, read *Mechanism_simulation_start.CATIAProduct* (or use the mechanism created above).

The additional mechanism is a cable pulling a lever to open a cover on the end. The idea is to create a simulation of how it is supposed to work, ie the cover opens and closes as the slider slides in and out. The slider is driven by the value of the angle of *Wheel* and the hinged cover is driven by the cable joint value.

Select *Simulation* icon and *Mechanism.1* in the window – OK

Two windows appear, one for varying the command values and the other for recording the positions.

Select *Insert* in the Edit window (recording the beginning status).

Using the *Command* window, change each command to new required values and select *Insert* again.

Repeat this for each required step in the simulation – note, when you *Compile* the simulation later, you can choose the number of intermediate steps which you want interpolated.

When you have inserted all the positions you require, select OK then *Compile Simulation* icon.

Fill in the required time step eg 0.2 then select OK – *Replay.1* will be created (which you can rename).

Select *Replay* icon and replay the motion.

You can look at a finished simulation by reading *Mechanism_simulation.CATIAProduct*.

15. CATIA Knowledge

Further information on CATIA Knowledge can be obtained using the online help. However here is a short summary of its functionality.

15.1 Parameters

When you create your CATIA document, a set of parameters (called intrinsic parameters) is automatically created corresponding to all the features defining the document. You can also create your own so called user parameters (there are 28 types of user parameters such as inertia, stiffness, mass, force, length, etc., which can have single or multiple values). The values of all these parameters can be derived from formulae, which may or may not contain some or all of these parameters. Hence for example a cylinder can be controlled to be twice as long as its diameter (select the $f(x)$ menu icon).

15.2 Rules

You can also create so called rules which consist of a series of instructions (programmed in CATIA syntax combined with Visual Basic) which also control the values of parameters and therefore may employ logic as well (select the *Rule* menu icon in *Knowledge Adviser* workbench).

15.3 Generative Knowledge

These rules can be extended to full document creation. Hence a program can replace all the interactive commands necessary to create parts, assemblies etc. combined with logic and algorithms which execute, perhaps taking information from a data base (select *Generative Knowledge* workbench).

15.4 Design Tables

Many variations of a part can be created by linking to a spreadsheet containing lists of different values for chosen parameters called a design table (select *Design Table* menu icon).

15.5 Optimisation

There are optimisation tools also which can be used, for example to minimise weight or area, (select *Product Engineering Optimise*).

For further help on these facilities,
in a CATIA session, select *Help/Contents, Index and Search*
In the help browser select *Product Synthesis* plus
Knowledge Adviser User Guide or
Generative Knowledge or
Expert Knowledge or
Product Engineering Optimise or
Knowledge Adviser Journaling Guide

15.6 A Simple Example of Knowledge Use

Before you start you need to check that the options are correct.

Select *Tools/Options*

Under *General/Parameters and measure* select all options above *design tables*.

Under *Infrastructure/Part Infrastructure/Display* select all specification tree options.

Under *Knowledge/ware/Product Functional Definition* select *Show Parameters* and *Show relations*

15.6.1 Creating Parameters, Formulae and Rules

Open the Part Design workbench in a new part and create a rectangular flat plate on the XY plane.

Create a through hole in one corner of the plate.

We will now create some parameters which will be used to control the part;

Select $f(x)$ icon and select *new parameter of type length*

Change the name to *Edge_distance* with 5mm default value.

Repeat this for *Hole_diameter* (default 5mm), *Plate_length* (default 100mm), *Plate_width* (default 50mm)

Create integer parameters *Number_of_holes_width* and *Number_of_holes_length* (default 2)

Create length parameter *Hole_spacing_width* then select *Add formula*

(Notice you have a window with *Hole_spacing_width* =)

We will now build the formula for this parameter.

Under *Dictionary* select *Parameters* and under *Members* select *Renamed parameters*
 Fill in by typing and selecting parameters from the list
 $(Plate_width - Number_holes_width * Hole_diameter - 2 * Edge_distance) / (Number_holes_width - 1)$
 OK.
 Repeat this for *Hole_spacing_length*.

Now we must use the parameters in the part.
 Double click the sketch for the plate and dimension the width and length.
 Double click the width dimension and right click its value, dragging to *Edit formula*.
 Using the same methods as before select *Plate_width*.
 Repeat this for plate length.
 Double click the hole and edit the sketch.
 Dimension the edge distances for the hole and set them to *Edge_distance* using the same method.
 In the hole creation window, right click the hole diameter and set that to *Hole_diameter*.

We need a hole pattern for multiple holes along width and length.
 Highlight the hole and select *pattern* icon.
 Under *First Direction*
 Select *Instances & spacing* and right click *Instances* then *Edit formula*
 Select *Number_of_holes_width*
 Right click *spacing* and *Edit formula* then create formula $Hole_spacing_width + Hole_diameter$.
 Repeat this for *Second Direction* and the length parameters.
 OK.

You should have one hole in each corner of the plate, you can edit the parameters in the tree and update to create different versions of the part. You may wish to control some of these parameters by using guidelines or rules. For instance you may want the hole spacing to be not less than a certain value and the hole diameter to be not less than another value. To do this select the *Knowledge Advisor* workbench.
 Select *Rule* icon – OK (You can fill in your own details first if you wish)
 Fill in $Hole_spacing_width > 3mm$ in the program area.
 In the message area fill in *Holes too close along width*.
 You can select *Silent*, *Information* or *Warning* for the type of check - in this case we will select *Warning*.
 Create another rule for the spacing along the length.
 Select *Rule* icon and create a rule $Hole_diameter > 2mm$ with appropriate message.
 Notice the message is triggered if the test is false.

The tree now contains these rules under *Relations* and are shown as traffic lights.
 Green is OK red if the rule has been disobeyed.
 Try changing the parameters to get the rules to be false.

Design intent, Company practice etc. can be invoked in a design in this way.
 You could create an empty part, intended to be used to create plates with holes, with all the parameters, formulae and rules already created but with no geometry. You then open this part and create the geometry setting the parameters accordingly.

15.6.2 Adding a Design Table

You can create an Excel spreadsheet containing various combinations of parameter values to give a family of parts.
 Select the *Design Table* icon (*Part Design* workbench, along the bottom menu, it may be under *fog* icon)
 Fill in name and comments as required and select *vertical* option - OK.
 Select *Renamed parameters* and select them all as required (you can use the shift key to select all at once).
 OK.

Now choose a filename and location for the Excel file. You get a window showing the current configuration.
 Select *Edit Table* in this window. You can do all the usual things with the Excel file which is opened, like reformatting columns and entering new rows of data.
 Save the Excel file and exit Excel. You get a synchronisation message in Catia and the new configurations are displayed. Close this message window.
 Double click a new row of values and update the model to get a new configuration – the rules still apply so will still warn of digressions. With the *Knowledge Advisor* workbench active you can double click the *Design Table* in the tree (under *relations*) to access it again.

15.6.3 Importing a Design Table

You could create a design table in Excel then import it to Catia.

Delete the design table from the tree (the Excel file still exists)
 Select *Design Table* icon and select *create design table from pre-existing file*.
 Select *vertical* option – OK
 Choose the Excel file to be imported.
 You get a message *automatically associate columns and parameters of same name?*
Yes – creates the table as before.
 If the input file has different column names you can choose *no* and manually associate the columns to the parameters.
 This Excel file could also contain other columns of data and calculations.

16. Capturing images

You can create a graphics file of a part using;

Tools/image/capture

This will capture the image as seen in normal display. If you want to create a realistic high quality image with texture and shadows use Studio Rendering.

17. Studio Rendering

Studio Rendering allows you to build an environment, add texture to your parts, add lights of various sorts and setup cameras. Realistic images of your product can then be created.

If you haven't already done so, you need to create a product and assemble the parts required. The environment, lights and cameras will be loaded into the same product.

If necessary, start CatiaV5 and select *Rendering* workbench (under *Infrastructure*).

This opens a new product which you could rename.

Select *Insert/existing component* (if necessary select the product in the tree)

Select *Aerotute_done_morphed.CATIAPart*

17.1 Adding texture to your surfaces

Expand *Part1* in the tree and double click *Surfaces*

Notice at the bottom of *Surfaces* there are *Plane* and *Window*, these are what we are going to render.

(Note, you do not have to apply material to get a rendered result – you will get the colour which the surface already has.)

Select *Apply material* icon (probably on the bottom row of icons)

Select *Metal/Aluminium*

Select *Plane*

Select *Apply material* – OK

Repeat this for *Window* using *Other/Plexiglass* as texture.

(Note this adds the material definitions to the tree. If you wish you can double click these and modify the definitions, thus modifying the rendered result.)

If you can't see the material texture in the graphics, you need to set the shading parameters.

To do this, select *View/Render Style/Customise View*

In the *custom view modes* window, select *shading* and *materials* (leave all other options unchecked) - OK

Select the green cylindrical shading icon to get the material shading on your part.

17.2 Creating an environment

If necessary, select the Product in the tree then the *Rendering* workbench.

Select *Create box environment* icon – a box is created surrounding the part and *Environment1* is added to the tree under *Applications*

Right click a chosen wall of the environment and drag to *properties*

Select texture tab and select a graphic file to be used (this can be a file created by another application or you can choose from a list of standard ones in *C:\ProgramFiles\Dassault Systemes\B07\Intel_a\startup\materials*)

(Another way of putting texture on the walls is to use the *Apply Material* icon as explained above.)

You can change the size of the environment box by selecting a wall and dragging it.

Create lights and camera using the icons, dragging to adjust position.

17.3 Creating the shot

When environment, camera and lights have been created, select *create shooting* icon.
 Fill in the camera, lights and environment in the *shooting definition* window – OK
 (Note you can leave out the camera creation and choose *current view* as the picture if you wish)

17.4 Taking the picture

Select *Render Shooting* icon.
 Select the *Camera* icon in the *render* window.
 This should give a rendered image.

You can modify other parameters in the *shooting definition* to get for instance a higher quality result.
 You can also choose ready made environments from the catalogue –
 Select *catalogue browser* icon and select *environments*.
 Select an environment to get a preview – double click to accept it. You can also modify an environment by right clicking the tree entry and dragging to properties.

18. Exporting Parts for Rapid Prototyping

In order to create a rapid prototype of your part you need to create an STL format file. Before you do this however you should decide on the orientation you want and the mesh approximation. Orientation can affect the quality of the result – the Actua 2000 rapid prototype machine in the School gives much better surface finish on top projection surfaces (z axis). So rotate your part with respect to the axis planes as required first.

Select *Rapid Prototyping* workbench
 Select *Tessellate an Object* icon
 Select the object
 Choose the approximation required
 Select *Apply* – OK

Select *Export* icon
 Select the tessellation
 Choose the file path and name (filetype .stl)
 OK

19. Making a Video

You can create a video of any Catia actions, the following explains how to make an AVI file from a mechanism.
 First create your mechanism and simulation as in sections 12 and 13.
 Next select *Tools/Image/Video*
 In *Video recorder window*, select *Setup* and in the *movie* tab select 25 frames/sec
 In the same tab select *compressor setup* and select *Microsoft MPEG-4 video code*, - OK
 You can also set the directory path and filename for the video file if you wish - OK.
 Now select the *Play a simulation* icon and select the simulation to be played in the tree.
 Turn off the tree using F3 (F3 again brings it back).
 In the *Player parameters* set the *sampling step* to 0.02 sec.
 Start the recorder in the recorder window and press play in the player window.
 When finished stop the recorder.
 You will have created an AVI file of your simulation.

APPENDIX

Available Tools for FADAL 3 Axis Machine and recommended machining conditions

(If you require a tool or work material not listed here, consult laboratory staff)

TOOL	* shows cutting/overall length			WORK MATERIAL					
	SHORT *mm	LONG *mm	CRNR RAD mm	STEEL			ALUMINIUM/BRASS		
				FEED mm/min	SPEED RPM	DEPTH mm	FEED mm/min	SPEED RPM	DEPTH mm
Dia 3mm End Mill (HSS)	9/50	15/60		800	2000	0.25	1200	6500	0.25
Dia 3mm Slot Drill (HSS)	6/50	15/60		800	2000	0.25	1200	6500	0.25
Dia 3mm Carbide	9/39			1000	6000	0.25	1500	10000	0.25
Dia 4mm End Mill (HSS)	12/60	20/60		800	2000	0.3	1200	5000	0.3
Dia 4mm Slot Drill (HSS)	8/60	20/65		800	2000	0.3	1200	5000	0.3
Dia 4mm Carbide	14/51			1000	6000	0.3	1500	10000	0.3
Dia 5mm End Mill (HSS)	15/60	25/65		900	1800	0.3	1200	4000	0.4
Dia 5mm Slot Drill (HSS)	10/60	25/65		900	1800	0.3	1200	4000	0.4
Dia 5mm Carbide	16/51			1000	5500	0.3	1500	10000	0.4
Dia 6mm End Mill (HSS)	15/60	25/65		1000	1700	0.3	1300	3350	0.5
Dia 6mm Slot Drill (HSS)	12/60	25/65		1000	1700	0.3	1300	3350	0.5
Dia 6mm Carbide	19/64			1200	5000	0.3	1600	10000	0.5
Dia 8mm End Mill (HSS)	20/65	35/80		1000	1250	0.3	1500	2500	0.5
Dia 8mm Slot Drill (HSS)	14/65	35/80		1000	1250	0.3	1500	2500	0.5
Dia 8mm Carbide	21/64			1400	3750	0.3	1600	7500	0.5
Dia 10mm End Mill (HSS)	25/75	45/95		900	1000	0.3	1600	2000	0.5
Dia 10mm Slot Drill (HSS)	18/70	45/90		900	1000	0.3	1600	2000	0.5
Dia 10mm Carbide	22/70			1600	3000	0.3	1800	6000	0.5
Dia 12mm End Mill (HSS)	30/80	55/105		800	850	0.3	1800	1700	0.5
Dia 12mm Slot Drill (HSS)	22/80	55/105		800	850	0.3	1800	1700	0.5
Dia 12mm Carbide	25/76			2000	3000	0.3	2000	5000	0.5
Dia 14mm End Mill (HSS)	35/90	55/110		700	750	0.3	1800	1450	0.5
Dia 14mm Slot Drill (HSS)	26/90	55/110		700	750	0.3	1800	1450	0.5
Dia 14mm Carbide	30/89			2000	2500	0.3	2000	4200	0.5

Dia 16mm	End Mill (HSS)	40/95	65/120		500	625	0.3	1800	1250	0.5
Dia 16mm	Slot Drill (HSS)	30/95	65/120		500	625	0.3	1800	1250	0.5
Dia 16mm	Carbide	32/89			1700	2500	0.3	2000	3750	0.5
Dia 20mm	End Mill (HSS)	45/110	75/140		400	500	0.3	1800	1000	0.5
Dia 20mm	Slot Drill (HSS)	40/110	75/140		400	500	0.3	1800	1000	0.5
Dia 20mm	Carbide	38/102			1400	2000	0.3	2500	3000	0.5
Dia 20mm	ballnose (carbide)				1400	2000	0.3	2500	3000	0.5
Dia 32mm	Endmill (carbide)				1200	2000	0.3	2500	3000	0.5
Dia 50mm	Endmill (carbide)				1000	1800	0.3	2000	2000	0.5

Available Tools and recommended machining conditions for the MAKINO FNC85 3 axis mill in lab. L110.

(If you require a tool or work material not listed here, consult laboratory staff)

TOOL TYPE	DIAM.	USEFUL LENGTH	CUTTING EDGE LENGTH	WORK MATERIAL			
				STEEL ROUGHING		ALUMINIUM ROUGHING	
				FEED MM/MIN	SPEED RPM	FEED MM/MIN	SPEED RPM
1 Carbide ball mill 1 tip	30	58	up to 20	100	1270	300	3800
2 Carbide end mill 4 tip	32	40	up to 10	400	1190	1200	3570
3 Carbide end mill 2 tip	50	90	up to 16	200	760	600	2280
4 Carbide end mill 5 tip	40	45	up to 10	500	950	1500	2850
5 Carbide end mill 4 tip	50	30	up to 12	400	760	1200	2280
6 Carbide end mill 20 tip	50	75	up to 20	400	950	1200	2850
7 Carbide face mill 8 tip	100	10	up to 10	200	380	600	1150
8 Carbide face mill 10 tip	170	10	up to 10	250	225	750	675
9 Carbide corner tool 2 tip	25	25	up to 1	400	1525	1200	4575
10 Carbide chanfer tool 2 tip	25	25	up to 4	400	1525	1200	4575
11 Carbide boring bar 1 tip	13 to 22	40	up to 1				
12 Carbide boring bar 1 tip	19 to 35	90	up to 1	See notes		See notes	
13 Carbide boring bar 1 tip	35 to 63	120	up to 1				
14 Carbide boring bar 1 tip	63 to 88	250	up to 1				
15 HSS face mill 10 tooth	100	150	up to 32	100	100	300	300
16 HSS face mill 10 tooth	75	250	up to 55	100	130	300	400
17 HSS end mill 6 tooth	25	90	up to 40	250	400	750	1200
18 HSS end mill 4 tooth	25	90	up to 40	150	400	450	1200
19 HSS ball mill 2 tooth	25	90	up to 40	80	400	250	1200

NOTES on Makino machining conditions

1. Cutting speeds for steel based upon surface speed of 120 m/min for carbide tipped tools
2. Cutting speeds for aluminium based upon surface speed of 360 m/min. for carbide tipped tools
3. For finishing increase speed by 50%.
4. For finishing reduce feed by 30%.
5. The above table can be used as a starting point. Depending upon the operations being performed, it may be possible to increase speeds and feeds or necessary to reduce speed and feeds.
6. For boring bars the speed/rpm to be used will depend upon the diameter of the boring bar setting. The feed rate is the maximum feed per revolution.
7. The column "CUTTING EDGE LENGTH" is the length of a tip or tooth that may be used for cutting. Depending upon the operation it may not be possible to machine to this depth in one cut. Generally, the maximum depth of cut should be limited to about 5mm.
8. Generally, use the largest tool possible for the operation required.
9. Tool numbers 15 and 16 are for soft materials only, eg. wood. DO NOT use on metals.
10. Use HSS (high speed steel) tools for machining wood or similar materials.
11. Generally, end mills and face mills cannot be used for plunge cutting. Obtain a slot-drill, or a tool designed for plunging, if plunging is to be performed..
12. ALWAYS, before finalising a machine program, check for the availability of the tool selected and have its size checked. Tools get broken, wear and are sometimes re-sharpened and hence their sizes may change. Go to the workshop or lab.
13. For harder/tougher materials speeds and feed should be reduced. For softer/less tough materials speeds and feeds can be increased.
14. The tools above may not be suitable for all operations, for example giving a bad finish.. Obtaining special inserts/tips may overcome difficulties.
15. CONSULT with lab./workshop for guidance upon the best tools and machining methods to use.

Available Tools and recommended machining conditions for the Deckel 5 axis mill.

(If you require a tool or work material not listed here, consult laboratory staff)

As on 22/01/2004

TOOL	* shows cutting/overall length			WORK MATERIAL					
	SHORT *mm	LONG *mm	BALLNOSE *mm	STEEL			ALUMINIUM/BRASS		
				FEED mm/min	SPEED RPM	DEPTH mm	FEED mm/min	SPEED RPM	DEPTH mm
Dia 3mm Carbide	-	-	10/38	3000	6000	0.25	4000	10000	0.5
Dia 4mm Carbide	14/51	-	14/50	3000	6000	0.25	4000	10000	0.5
Dia 5mm Carbide	-	-	16/52	3000	5000	0.25	4000	10000	0.5
Dia 6mm Carbide	-	-	19/63	3000	5000	0.25	4000	8500	0.5
Dia 8mm Carbide	-	-	21/62	3500	4000	0.25	4500	6500	0.5
Dia 10mm Carbide	22/70	-	25/70	3500	4000	0.25	4500	5000	0.5
Dia 12mm Carbide	25/76	-	25/75	3500	4000	0.25	4000	4000	0.5
Dia 20mm Carbide	38/102	-	-	3000	3000	0.25	4000	3500	0.5
Dia 40mm Carbide	40/80	-	-	2500	2000	0.25	4000	3000	0.5