# TABLE OF CONTENTS

Introduction ..............................................................1
Drafting ...........................................................2
Drawing Screen .....................................................3
Pull-down Menus ....................................................4
  File ......................................................................4
  Edit ....................................................................5
  View ...................................................................6
  Insert ..................................................................7
  Tools ...............................................................8
Drafting Workbench .....................................................9
  Views and Sheets ................................................9
  Dimensions and Annotations ................................10
  Drawing tools ....................................................11
  Additional options .............................................13
Bottom Toolbar Changes ..............................................14
Top Toolbar .......................................................15
  Text ..................................................................15
  Dimension .......................................................16
  Graphic properties ............................................16

Drafting Basics ...........................................................17
  Starting a New Drawing ........................................17

Creating Views from a Part .............................................23
  Front View .......................................................24
  Orientation Circle ...............................................25
  Projection View ..................................................31
  Isometric View ..................................................34
  Advanced Front View ...........................................35
  Local Axis System ..............................................37
  Unfolded View ...................................................38
  Extracted View from 3D .........................................40
  Auxiliary View ..................................................41
  Section Views and Section Cuts ...............................43
  Detail Views ........................................................49
  Clipping Views .....................................................54
  Broken View .....................................................55
  Breakout View ...................................................57
  Configuration of Views .........................................60
  Review .............................................................67

Modifying Sheets ........................................................72
  Sheet Properties ..................................................72

Page Setup ....................................................................73
### CATIA Drafting

**Modifying Views** ......................................................... 74  
  View Properties .......................................................... 74  
  View positioning ......................................................... 77  
  Locating Views ............................................................ 80  
  View Names ................................................................. 80  
  Restore Deleted .......................................................... 81  
  Updating Views ............................................................ 82  
  Show/NoShow .............................................................. 84  
  Callout properties ....................................................... 85  
  Callout Definition ....................................................... 87  
  Unbreak and Unclip ..................................................... 89  
  Modifying Projection Plane ........................................... 90

**Dimensioning** ............................................................ 93  
  Creating Dimensions .................................................... 93  
    Length/distance ....................................................... 93  
    Angle ........................................................................ 101  
    Radius ........................................................................ 102  
    Diameter ..................................................................... 103  
    General dimension ..................................................... 104  
    Dimensions with Intersection points ................................ 107  
    Chamfer ...................................................................... 108  
    Thread ........................................................................ 110  
    Coordinate ................................................................... 111  
    Hole dimension table .................................................. 112  
    Coordinate dimension table ......................................... 115  
    Chained ...................................................................... 117  
    Cumulated .................................................................... 118  
    Stacked ........................................................................ 119  
    Fillet Radius Dimensions ........................................... 120  
    Dimensions for curves .................................................. 121  
    Setup Parameters ......................................................... 126  
    Geometrical Dimensioning and Tolerancing ...................... 133

**Modifying Dimensions** .................................................. 141  
  Top Toolbar ................................................................. 141  
  Pull Down Menu Tools, Options ....................................... 142  
  Geometrical Tolerance .................................................... 144  
  Properties ...................................................................... 145  
  Analysis ........................................................................ 160  
  Positioning .................................................................... 162  
  Interruptionsm ............................................................... 165  
  Update .......................................................................... 167

**Generating Dimensions** .................................................. 169
### Table of Contents

**Annotations** ............................................................ 183
- Creating Text .......................................................... 183
- Modifying Text .......................................................... 187
  - Top Toolbar .......................................................... 187
  - Adding a Leader .................................................... 201
  - Orientation Link ................................................... 206
  - Positional Link ..................................................... 207
  - Attribute link ..................................................... 208
  - Replicate Text ..................................................... 209
  - Query Object Links ................................................. 210
  - Isolate Text ....................................................... 211
  - Element Positioning .............................................. 212

**Tables** ........................................................................ 213
- Creating Symbols .......................................................... 221
  - Balloon ............................................................... 221
  - Roughness Symbol .................................................... 223
  - Welding Symbol ...................................................... 225
  - Weld ............................................................... 227

**Markup** ........................................................................ 228
- Center lines and Axis lines .............................................. 228
- Area Fill ................................................................. 231
- Modifying an Area Fill .................................................... 234
  - Properties ............................................................. 234
  - Top Toolbar .......................................................... 237
- Graphical properties ...................................................... 239
- Arrows ........................................................................ 240

**Assembly** ..................................................................... 241
- Creating an isometric view .................................................. 241
- Creating a view from a scene ............................................. 242
- Generate balloons ........................................................... 244
- Bill of Material ............................................................ 245
- Other views ................................................................. 246
- Properties of a part in an assembly ....................................... 248
- Overload properties ....................................................... 250
- Cut, Copy and Paste views .................................................. 253
- Views of parts of an assembly ............................................ 255
- View Links ................................................................. 258

**2D Geometry** ............................................................. 261
- Creating a view ............................................................ 261
- Drawing tools .............................................................. 261
- View plane definition ...................................................... 265
- Multiple view projection .................................................... 266
- 2D Components ........................................................... 269
- Background ............................................................... 274
# Table of Contents

**Review** .......................... 281  
  Geometry in all viewpoints .......................... 286  
  Filter generated elements .......................... 286  
  Exporting ........................................ 286  

**Problems** .......................... 287  
  Problem #01 ..................................... 287  
  Problem #02 ..................................... 288  
  Problem #03 ..................................... 289  
  Problem #04 ..................................... 290  
  Problem #05 ..................................... 291  
  Problem #06 ..................................... 292  
  Problem #07 ..................................... 293  
  Problem #08 ..................................... 294  
  Problem #09 ..................................... 295  
  Problem #10 ..................................... 296  
  Problem #11 ..................................... 297  
  Problem #12 ..................................... 298  
  Problem #13 ..................................... 299  
  Problem #14 ..................................... 300  
  Problem #15 ..................................... 301  

**Appendix A** .......................... 303  
  Mechanical Design - Drafting - General ............... 303  
  Mechanical Design - Drafting - Layout ............... 304  
  Mechanical Design - Drafting - View .................. 305  
  Mechanical Design - Drafting - Generation .......... 306  
  Mechanical Design - Drafting - Geometry ............. 307  
  Mechanical Design - Drafting - Dimension ............ 308  
  Mechanical Design - Drafting - Manipulators ........ 310  
  Mechanical Design - Drafting - Annotation and Dress-Up 311  
  Mechanical Design - Drafting - Administration ...... 312
Creating Views from a Part

There are many types of views that can be created in CATIA Drafting. This section is designed to cover all of the various types of views that can be created. As discussed earlier you can create a drawing in one of two ways, you can either begin a drawing with an empty sheet or with a configuration of views. Initially you will be starting a new drawing with an empty sheet.

Before you begin, make sure that you do not have any other windows open in CATIA.

Open the Views document. You should notice that this is a part. You will be creating views for this part.

Start a new drawing with an empty sheet, the settings should be ASME Standard, D ANSI Sheet Style and Landscape orientation. You do not want any views in the initial drawing. You can always assume ASME Standard and Landscape orientation for all exercises unless otherwise specified.

Change your window configuration to be Tile Horizontally. You can do this by using the pull down menu Window, Tile Horizontally. It should appear similar to the diagram shown below.
Front View

The first view created is normally the front view. You can create a front view by defining the 3D object that you want to use and defining the plane that should be used to define the view.

Select the front view icon. This will allow you to define a plane from a 3D object and create the front view.

Select the plane shown below from the part. This will define the plane that you want to use for the front view.

The diagram shown below should appear in your drawing window.

Notice that the part appears in the drawing window and you have a blue orientation circle in the upper right-hand corner of your window. This circle allows you to orient your view before actually creating it. Once the view is oriented the way you want it then you can either select on the center dot or select outside the blue circle to create the view. You are going to investigate the various options that you have available for orienting the view.
Orientation Circle

You can select on the right or left arrows to rotate your part inside the view to give you a different plane in which to create your front view. The same goes for the up and down arrows but it rotates your part in the other direction. This will rotate the part 90 degrees with each selection. The two curved arrows will rotate your part within this plane either clockwise or counterclockwise. The amount that it rotates defaults to 30 degrees but you can edit the angle. You can also use the green knob to rotate your part around by selecting it and dragging it around the circle, it defaults at 30 degrees but it can be edited as well.

Select the right arrow until you turn it all the way around back to the original orientation. You should have to select the right arrow four times. The diagrams below show what your part should look like after each selection.

![Diagrams showing part rotations](image)

Select the up arrow until you turn it all the way around back to the original orientation. You should have to select the up arrow four times. The diagrams below show what your part should look like after each selection.

![Diagrams showing part rotations](image)
Select the right curved arrow three times. You should notice the green knob now turning around the circle as well as the part turning. The diagrams below show what your part should look like after each selection.

![Diagram showing part rotations](image1)

Notice that it rotated the part 30 degrees each time. You can modify that by selecting the third mouse button while on the curved arrow and choose the Edit Angle option.

Using the third mouse button select on the right curved arrow and choose the Edit Angle option. The Angle window should appear.

![Angle window](image2)

Change the Rotating angle to 45 degrees and select OK. Now when you select on the curved arrows they will rotate the part 45 degrees instead of 30 degrees.

Select the left curved arrow two times. This should turn your part back to its original orientation.

You can also use the green knob to rotate your part. All you have to do is select the green knob and while holding the button down, drag it around the circle. This also defaults to 30 degree increments. You can change the settings for how the green knob works by using the third mouse button. You have to be very careful when trying to select the green knob because if you select off of it and outside the blue circle the view will be created automatically. You are now going to change the angle of your part to be at 90 degrees.
Using the third mouse button select on the green knob. Make sure you are on the green knob when you press the button. This will give you the following options.

- **Free hand rotation** Allows you to rotate the part at any angle
- **Incremental hand rotation** Allows you to rotate the part at a given increment
- **Set increment** Sets the amount of the increment for *Incremental hand rotation*

**Set current angle to** Allows you to specify an angle that you want to rotate the part

Select the *Set current angle to* option and choose 90 deg from the menu. This rotates your part to be at 90 degrees. You are now ready to create the actual view since you have it oriented the way you want.
Select on the center dot or select outside the blue circle. The front view is created and should appear similar to the diagram shown below.

![Front View Diagram]

Notice that the frame, dashed box, of the view changed from green to red. The green signifies that it is the view that is getting ready to be created. The red signifies that the view is the active view. You will now look at the properties of the view.

**Maximize your drawing window.** This will allow you more room to work.
Using the third mouse button select the Front view from the specification tree and choose Properties. Notice that it is underlined, this also signifies that it is the active view.

**Visualization and Behavior**

- **Display view frame**
  Toggles the display of the view frame

- **Lock View**
  Locks the settings so they can not be changed

- **Visual Clipping**
  Allows you to display just a portion of the view

**Scale and Orientation**

- **Angle**
  Sets the orientation angle of the view

- **Scale**
  Sets the scale of the view

**Dressup**

All options toggle if you want to see those elements in the view or not and how those elements will be displayed
View Name  Allows you to specify a name for the view or the 2D component when valid

**Generation Mode**

- **Only generate parts larger than**  Specifies the minimum size a part can be and still be generated
- **Enable occlusion culling**  Allows you to turn on occlusion culling which will only load the parts that are used in that view instead of loading all of them
- **View generation mode**  Changes how the view is generated, *Exact* is all of the geometry, *CGR* is the external appearance of the object and *Raster* is an image

**Generative view style**  Defines the style that was used to generate this view, this option is only available if you have turned off the Prevent generative view style creation option under pull down menu Tools, Options

You will work more with the properties as you develop more views.

**Select Close.**  This will return you to the drawing.  You are now ready to create some more views.
Projection View

This view is created by using an existing view, usually the front view, and then selecting a position for the view. The view that gets created depends on where you select. If you select to the left of the view then you get the left view. Projection views are created orthogonally to the active view.

**Select the projection view icon.** It is located under the front view icon. You can now position the cursor and select the view that you want to create.

**Position the cursor to the left of the front view and select.** You should have seen a preview of the part before you selected. Your drawing should appear similar to the diagram shown below.

![Diagram of projection view](image)

A closer look at the view is shown below.

![Closer look at projection view](image)

The blue frame means that it is not the active view.
Select the projection view icon again and create the top view. Your drawing should appear similar to the diagram shown below.

A closer look at the view is shown below.

You can move the views around within the sheet. The default setting is for the views to align with each other. Therefore you can only move the left view right and left and you can only move the top view up or down. You move the view by selecting on the frame of the view with the first mouse button and while holding the button down, drag the view with the mouse. If you move the front view around, all the other views will move with it. You will learn how to make the views not align later in this course.
Move the views around so that it appears similar to the diagram shown below.
Isometric View

To create an isometric view you orient your part in Part Design the way you want it and then using the appropriate icon you select the body and the isometric view is generated.

Change the configuration of your windows to *Tile Horizontally*.

**Select the isometric view icon.** It is located under the front view or projection view icon. This will allow you to select the part to create an isometric view.

**Select the part from the Part window.** The preview of the isometric view appears in your drawing. Once again you can use the orientation circle to orient the view in what ever position you like.

**Select outside the blue circle to create the view.** It should appear similar to the diagram shown below. *Note: only the view is shown below.*

Move the isometric view so that your drawing appears similar to the one shown below.

Save your drawing and close all documents.
Advanced Front View

The advanced front view option works exactly the same as the front view option except that you can specify additional information during the creation of the view. Examples of the information that can be specified are the view name and view scale.

Open the Advanced Front View document. You are going to create a new drawing using the advanced front view option.

Start a new drawing with an empty sheet, the settings should be ASME Standard, D ANSI Sheet Style and Landscape orientation. You do not want any views in the initial drawing. You can always assume ASME Standard and Landscape orientation for all exercises unless otherwise specified.

Change the configuration of your windows to Tile Horizontally.

Select the advanced front view icon. It is located under the front view or isometric view icon. A View Parameters window appears. You can specify the View name and the Scale for the view before you create the actual view.

Change the View name to be Large View and the Scale to be 2.0 and select OK.

Select the face shown below.
Create the view. You can create the view by selecting on the sheet or by selecting the center of the orientation circle. It should appear similar to the diagram shown below.

You should notice that the name of the view is Large View and it has a scale of 2:1 just like you specified. You will now create a front view using a local axis system.
Local Axis System

When you create a view from your part, you have the option to select an axis system that you want to use as the view axis. If you do not select an axis system before selecting the part then it will use the global axis as the view origin.

Select the front view icon. You are going to select an axis system first and then define the plane for orientation.

Select Axis System.1 from the specification tree in the part and then select the same face as before. You will probably have to expand the Axis Systems branch in order to see the actual axis system.

Create the view. The view is created similar to the diagram shown below.

Notice that the view axis is in the location of the local axis system from the part instead of using the global axis position.

Save your drawing and close all documents.
Unfolded View

The unfolded view option is used to create a view of a sheetmetal part created in CATIA. It is defined similar to the front view but it unfolds the view for you onto the drawing. This option only works with parts created using the sheetmetal options available in CATIA.

Open the Unfolded View document. You should notice that this is a part with Sheet Metal Parameters. You will be creating an unfolded view of this part.

Start a new drawing with an empty sheet, the settings should be ASME Standard, D ANSI Sheet Style, Landscape orientation. You do not want any views in the initial drawing. You can always assume ASME Standard, Landscape orientation for all exercises unless otherwise specified.

Change your window configuration to be Tile Horizontally. You can do this by using the pull down menu Window, Tile Horizontally.

Select the unfolded view icon. It is located under the front view or advanced front view icon. This will allow you to select a face of the part to create an unfolded view.

Select the face of the part shown above. The preview of the unfolded view appears in your drawing.
Select outside the view to create the view. It should appear similar to the diagram shown below. Note: only the view is shown below.

If the axis lines do not appear in your view, change the Properties of the view to show Axis under Dress Up. Notice that the view shows the sheetmetal part as if it was unfolded. This will give you an accurate view of the actual piece of sheetmetal that would be required to bend into the desired shape.

Note: This is very different than creating a front view of this part using the same face. The diagram below shows you what the front view would have looked like.

It is important to remember that this option only works if the part was created with the sheetmetal options.

Save your drawing and close all documents.
Extracted View from 3D

The extracted view option will create a view extracted from a part that has a view defined. It is defined by selecting a view that was created in other operations. This exercise will show you how to use this option with a view generated for functional dimensioning and tolerancing.

Open the Extracted View document. You should notice that this is a part with an Annotation Set that includes a Front View. You will be extracting a view from this FD&T view.

Start a new drawing with an empty sheet, the settings should be ASME Standard, D ANSI Sheet Style, Landscape orientation. You do not want any views in the initial drawing. You can always assume ASME Standard, Landscape orientation for all exercises unless otherwise specified.

Change your window configuration to be Tile Horizontally. You can do this by using the pull down menu Window, Tile Horizontally.

Select the view from 3D icon. It is located under the front view or unfolded view icon. This will allow you to select a FD&T view from the part.

Select the FrontView from the specification tree of the part. It should show you a preview of the view in the drawing window. It is important to note that the view created in FD&T should use the same standard as the sheet that is defined.

Select outside the view to create the view. It should appear similar to the diagram shown below. Note: only the view is shown below.

It is important to remember that this option only works with parts that have views defined from a previous operation such as functional dimensioning and tolerancing.

Save your drawing and close all documents.
**Auxiliary View**

The last projection view that will be covered is the auxiliary view. This is used to create a view that is not orthogonal to an existing view but is looking at a particular side of the part. It is defined by selecting an element that defines a view plane in the current view.

**Open the Auxiliary View drawing.** Make sure you open the CATDrawing and not the CATPart. You should see a drawing that contains a *Front view* and a *Top view*.

You are going to create a view looking at the angled side of the *Front view* as shown above.

**Select the auxiliary view icon.** It is located under the front view or view from 3D icon. This will allow you to select an existing line to define the orientation of the view or you can define a start point and end point to define your own line. Either way it will create a view looking at your part normal to the defined line.

**Select the line shown above from the Front view.** This will define your auxiliary view to look at the slanted portion of the part. You now need to define where you want the auxiliary callout to be positioned.
Select somewhere to the right of the angled line. Now you can define the position for your view.

Select to the right of your auxiliary callout. The view should appear similar to the diagram shown below.

You can always reposition your auxiliary view by selecting on the frame and moving it. This method used an existing line but it is important to remember that you can define your own line by selecting a start point and an end point.

Save your drawing and close all documents.
Section Views and Section Cuts

Section views and section cuts are views that show the profile of a part at a position, with the areas that contain material being filled with a pattern. If the part has a material applied to it then the pattern will display based upon that material. The difference between a section view and a section cut is that a section view will show what geometry lies beyond the cut line whereas a section cut will only show what exists at the cut line. The cut line can be either a single line or it may have jogs in it to show a cut through the same part at different places. Also available are aligned section views and aligned section cuts. The aligned style allows there to be cut lines that are not parallel and the resulting view is shown perpendicular to the cut lines.

Open the Section Views and Cuts drawing. Make sure you open the CATDrawing and not the CATPart. You should notice that the Front view is active since the outline is red. You are going to want the Top view to be active since you are going to create a section view using the Top view.

Double select on the Top view. The frame of the Top view should change to red. Before you define the cut line you will want to open the Section Views and Cuts part.

Open the Section Views and Cuts part and Tile Horizontally. You do not have to have this part opened in order to create a section view but it will show you the cut plane in the 3D part as you define the cut line.

Select the offset section view icon. This will allow you to define a cut line in your top view.

Select to the first location to define the beginning of your section cut as shown below.

Move your mouse over to the second location as shown above but do not select the location.
You should notice the cut plane appears in the part window as you are defining the cut line. It should appear similar to the diagram shown below.

**Double select at the second location.** Since this is the end of your cut line you must double select. You can now position your section view.

**Position the view above the Top view as shown below.** You should notice the section arrows appear on your cut line.

*Note: You also have the option of selecting circular edges to define the cut lines. If you select on a circle it will automatically use its center point as the cut line location.*
You are now going to create a section cut.

**Select the offset section cut icon.** It is located under the offset section view icon. This will allow you to define cut lines but instead of generating a section view it will generate a section cut.

**Define your cut lines as shown below.** The numbers show the order of selection. Make sure you double select your last location.

Position your view to the right of the *Top view* as shown below.
You should notice that the section cut does not show any geometry other than what the cut line actually touches. That is the difference between a section cut and a section view.

**Make the Front view active.** You can do this by double selecting on the *Front view*. You are going to create an aligned section view using the *Front view*.

**Select the aligned section view icon.** It is located under the offset section view or offset section cut icon. With this option you can create cut lines that are not parallel.

**Define your cut lines as shown below.** The numbers show the order of selection. Make sure you double select your last location.

Position the view down and to the left of the *Front view*. The view is positioned normal to one of the cut lines.
Change the *Properties* of the aligned section view to not show *Hidden Lines*. It should appear similar to the diagram shown below.

You have the option of creating aligned section views or cuts, however this exercise only shows you the option of creating an aligned section view. Creating an aligned section cut is done in the same manner except you use the aligned section cut icon. Feel free to try the option out on your own.

The next item that will be discussed in terms of section views and cuts is that you have the option of using a planar surface or an actual plane from your 3D part to define the cut line.

*Make sure you have the part window and the drawing displayed with *Tile Horizontally*. This will enable you to see what is going on in both windows. Make sure the *Front View* is still active.

*Create a plane in your part that is offset from the xy plane 0.25 inches.*

*Go back to your drawing and select the offset section view icon.* You may have to update your views since you made a change to your part. If you do just select the update icon.

*Select the created plane from your part.* Notice the cut line automatically appears in your *Front view*. The cut line is not associated or linked to the plane and can be modified in the drafting workbench.
Position the view under the *Front view* as shown below.

You can also create a sketch in your part that can be used to define a section line. When you use a sketch to define a section line, the section line is associated or linked to the sketch. If you want to modify the section line you would need to open the part and modify the sketch and then update your views. If you erase the sketch that was used to define the section line, then the section line would no longer be linked and it would be converted to a normal section line that could be modified within the drafting workbench.

**Save your drawing and close all documents.**
Detail Views

A detail view is a partial view of a part that is usually at a higher scale to make it easier to see. CATIA allows for the creation of four different detail views. You can either use a circle or a profile callout. Besides choosing what you want to use as a callout, you can either create the detail view so that it uses the 2D projection to determine the detail view or it can generate the view from the 3D definition. The quick detail options use the 2D projection to generate the view.

Open the Detail Views drawing. Make sure you open the CATDrawing and not the CATPart. You should notice that the Front view is active since the outline is red. You are going to create a couple of detail views to draw attention to various areas that may be hard to understand with the current views.

Select the quick detail view icon. It is located under the detail view icon. This will allow you to define a circular callout to define the detail view. Since you are using the quick option this will utilize the 2D projection to define the detail view. A Tools Palette window appears. This window appears on a variety of icons and will show the available options for the icon selected. In this case you can specify an exact radius for the callout. You are going to define the radius by selecting a location instead of keying in a value.

Select the center of your circle and define your radius as shown below. This will define the area that is used in the detail view. A preview of the area of the view should appear for you to position on your sheet.
Position the detail view to the right of the Front view. You should see the view appear similar to the one shown below. Also you should note that view defaulted to a scale of 2:1. You can adjust this scale by changing the properties of the view.

Notice that the view name corresponds with the letter assigned to your circle callout. This was created using the quick detail option. Next, you will create a detail view not using the quick option.

Make the rightmost isometric view active. You can do this by double selecting on the view.

Select the detail view profile icon. It is located under the detail view or quick detail view icon. This option will let you use a profile to define the callout. You can do this by selecting the endpoints of lines defining the profile. You can either close the profile yourself or you can double select to end the profile definition. If you double select, the profile will close automatically using a line connecting the last and first point defined.

Define the profile as shown below. The arrows point to the corresponding endpoints.
Position the view to the right of the isometric view. It should appear similar to the diagram shown below.

Notice that this view does not show the profile line past the edges of the part, in other words, the detail view is trimmed to the part edges. This is because the view is using the 3D definition to define the view instead of the 2D projection.

You will now create two other detail views to better illustrate the difference between the quick detail view and the regular detail view.

Make the Bottom view active.

Select the detail view icon. It is located under the detail view profile icon. This will allow you to define a circular callout. The Tools Palette window appears.

Define a circle as shown below.
Position the detail view to the right of the Detail B view. It should appear similar to the diagram shown below. Once again, notice that the view is trimmed to the edges of your part. Unlike the quick detail option that you used the first time which shows the entire circle.

![Diagram of Detail B view](image)

Make the leftmost isometric view active.

Select the quick detail view profile icon. It is located under the detail view icon. This will allow you to define a profile to use as the callout.

Define a profile similar to the one shown below.

![Diagram of Isometric view](image)
Position the view above the Detail C view. You should notice that the profile definition does not stop at the edges of the part, instead the entire profile is shown. This is because the quick options use the 2D projection and not the 3D definition.

Hopefully this exercise gives you a good example of the difference between the various detail view options.

Save your drawing and close all documents.
Clipping Views

Clipping views are similar to detail views in the way they can be defined. You have the option of either using a circle or a profile to define the clipping area just like the detail view. However the difference is that a clipping view actually changes the current view to just contain the clipping area. This option is used when the whole part would be too large and you are only interested in a section of the part. By default the clipped view will maintain its scale ratio.

Open the Clipping Views drawing. Make sure you open the CATDrawing and not the CATPart. You should notice that the Auxiliary view is active since the outline is red.

Select the clipping view profile icon. It is located under the clipping view icon. You are going to define a profile that you want the auxiliary view to contain instead of it showing the entire part.

Define the profile as shown below. This is done the same way as you defined a profile for a detail view.

Notice that the view automatically changes after you finish defining the profile.

It is important to understand that this did not create a new view but changed the existing view to only show the clipped area. The clipping view option will do the same thing except you define a circular area instead of using a profile.

Save your drawing and close all of the documents.
Broken View

A broken view is used when you have a long section that is not important to be seen in the view. This option will allow you to define the two ends of the break and then it will modify the view so that section will not appear. Breakout lines are generated to show where the breaks occur.

Open the Broken View drawing. Make sure you open the CATDrawing and not the CATPart. You should notice that the Front view is active and that the Left view extends past the sheet. You are going to use the broken view option to modify the Left view to show the two ends of the beam leaving out the middle section.

Make the Left view active.

Select the broken view icon. You are going to define two break lines that will define the area to remove when modifying the view.

Select inside the beam edges at the location shown below. Once you select the location you can either create the broken view to break horizontally or vertically. In this case you want the break to be vertical.

Select above the previous location. You will now be able to define the other break line so that it can remove the area between them.

Select inside the beam edges at the location shown below.

Select outside the view on the sheet. The broken view is generated. However the view is still partially off the sheet and the label is to the right of the actual geometry.
Drag the text underneath the geometry. You can drag the text if you hold the first mouse button down while on the text.

Drag the view over to the right until it is completely on the sheet. You can drag the view by holding the first mouse button down on the frame of the view. The finished drawing should look similar to the one shown below.

You can define more than one break in a view but it has to be in the same direction. That would be useful if there was a pocket removed out of the middle of the beam and you wanted to break the view in order to show the two ends and the pocket definition.

Save your drawing and close all documents.
Breakout View

A breakout view is used when you have a current view that needs to have a section removed in order to better see the definition that is behind the section. Hidden lines could be used to show the definition but sometimes it is not very clear because of the number of hidden lines. You can define a profile of the section that you want removed in the view and then you have the option of using a 3D viewer to define the depth of removal using another view. This will modify the existing view to show the breakout.

**Open the Breakout View drawing.** Make sure you open the *CATDrawing* and not the *CATProduct*. You should notice that the *Front view* is active. There are two top views, one with hidden lines and one without. The one with the hidden lines is a little congested because of the hidden lines of the bracket. In this particular case you are more concerned with what the top view looks like without the top of the bracket.

**Make the left Top view active.**

**Select the breakout view icon.** It is located under the broken view icon. You are going to define the break out area to remove when modifying the view.

**Define the profile as shown below.** The *Tools Palette* window appears allowing you to specify the length and angle of each linear section. You can define this profile exactly the same way as you defined a profile using the detail or clipping option.
After you define the profile a 3D Viewer window should appear.

The 3D Viewer allows you to see a preview of the depth of the removal and you can use the orange line to define the depth of the cut. If you use the line in the 3D Viewer it is not linked to any geometry and is only an approximate location. You can also define the depth by selecting an element in another view and it will be linked to that element. In addition, you can specify a Reference element and a Depth using the available options.

**Animate**

Allows the 3D Viewer window to change as you position your cursor in another view.

**Depth Definition**

**Reference element**

Specifies the element that you want to use as the depth and you have the additional option of defining a Depth from that element.

Select the **Animate option in the 3D Viewer window.**

**Move the cursor to the Front View.** Notice that the image changes to show the Front view orientation.
Select the bottom dashed line of the bracket as shown below. This will define the depth of the breakout.

Select OK. Even though the position may change as you move the cursor to the OK button it should keep the selection specified. The breakout view takes affect and the Top view changes. It should appear similar to the diagram shown below.

This option can be very useful to show an area of internal material, rather than using hidden lines. You should notice that this option only modifies an existing view just like the broken view and clipping view options.

Save your drawing and close all documents.

This finishes the discussion on creating views individually. You will now investigate the configuration options available.
Other available courses

CATIA V5 and ENOVIA

• CATIA Basic Concepts
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• CATIA Assembly Design
• CATIA Drafting
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• CATIA Prismatic Machining
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