

A large, light blue wireframe sphere is positioned on the left side of the page. It is composed of numerous thin lines that form a grid of latitude and longitude, giving it a three-dimensional appearance. The sphere is centered vertically and horizontally on the left half of the page.

**AVEVA**

MARINE

# Catview User Guide

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# Catview User Guide

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# 1 Introduction

Catview is a freeware application for building and maintaining piping specifications. It is designed to be largely independent of catalogue naming conventions, but requires some form of catalogue organisation for it to work.

The principle features of Catview are as follows:

- Catalogue Listing and Browsing.
  - Display a list of component types using the category description attribute.
  - Sort the catalogue by generic type and standard.
  - Display data sheets showing catalogue items and their parameters.
  - Search the catalogue for specific words or phrases.
- Specification Building using the Browsing functionality.
  - Create new piping specifications.
  - Build components into Specifications using the browser to select appropriate component types.
  - Add a list of items using size ranges.
  - Ignore non-preferred sizes using an exclusion list.
- Specification Editing.
  - Display detailed list of Specification components sorted by Component Type.
  - Edit component details such as Name, Stype, Material, Component Reference, Bolt Reference and Description.
  - Add wall thickness details.
- Specification Reporting Automatic Spec Summaries.
  - PEGS Specification Transfer.
- Catalogue Documentation.
  - Produce Catalogue datasheets via OUTFITTING DRAFT.

## 1.1 Catview Contents

Catview is a OUTFITTING application, which works with the Standard OUTFITTING master project MAS, as supplied with all OUTFITTING software releases. The Catview release contains the application directories, a set of category datasheets in OUTFITTING PLOT file format and a OUTFITTING DRAFT Macro directory for building new data sheets.



## 2 Installation

### To install Catview

- Insert the CD, into your CD drive.
- Wait for the CD autorun program to show the Cadcentre startup screen.
- Select the index link to list the CD contents.
- Select the Catview Product
- Follow the instructions from the install shield program

**Note:** The default directory for installing Catview is C:\CADCENTRE\Catview12.0. If you choose to install the application somewhere else, you will need to edit the evars.bat file inside the Catview directory to reflect the new location.

To do this, the line “**set Catview=c:\cadcentre\Catview12.0**” will need to be changed to point to the correct directory

### To add Catview to the OUTFITTING applications

- Edit the standard evars.bat file in your OUTFITTING directory to add a line to call the Catview evars.bat file.
- Assuming you have a standard OUTFITTING installation, the following line should be inserted at the end of the evars.bat file:

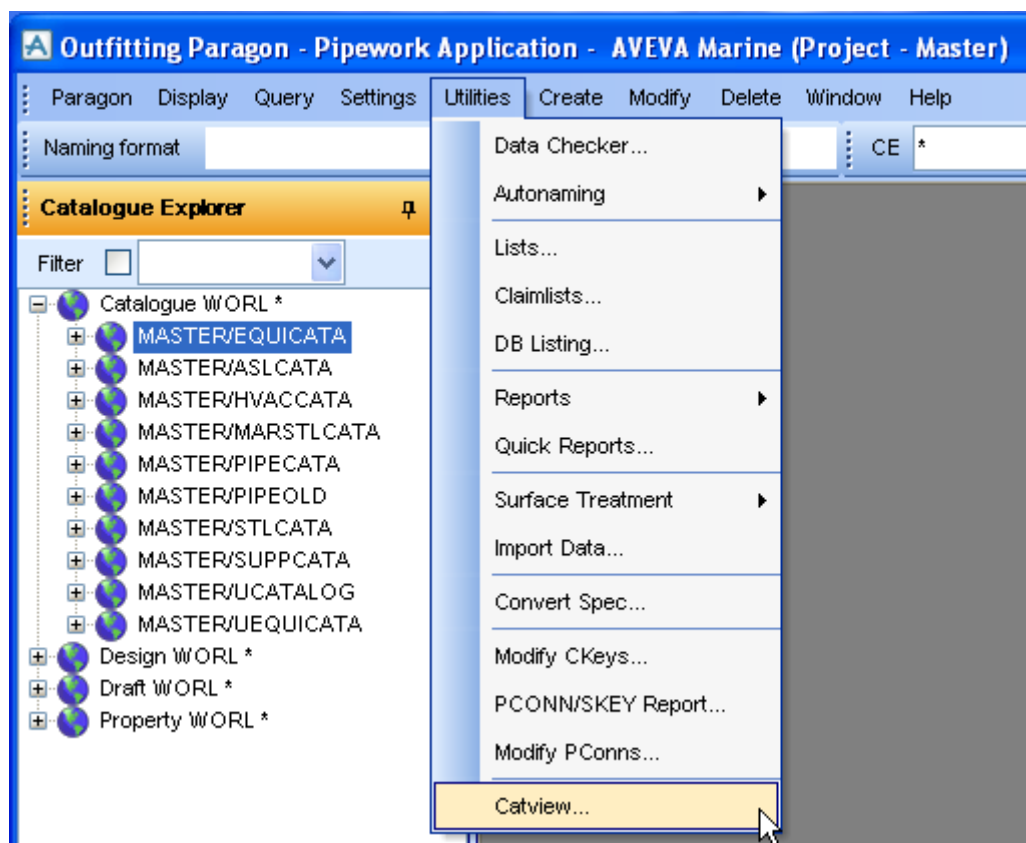
```
call C:\cadcentre\Catview12.0\evars.bat
```



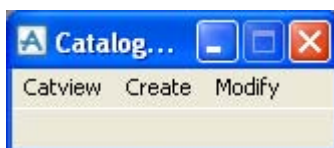
### 3 Starting Catview

To use Catview, start OUTFITTING and enter the AVEVA MAS project in PARAGON.

When installed, Catview is accessible from the PARAGON piping application under the **Utilities>Catview** menu as shown:



This loads the Catview main menu as shown below:

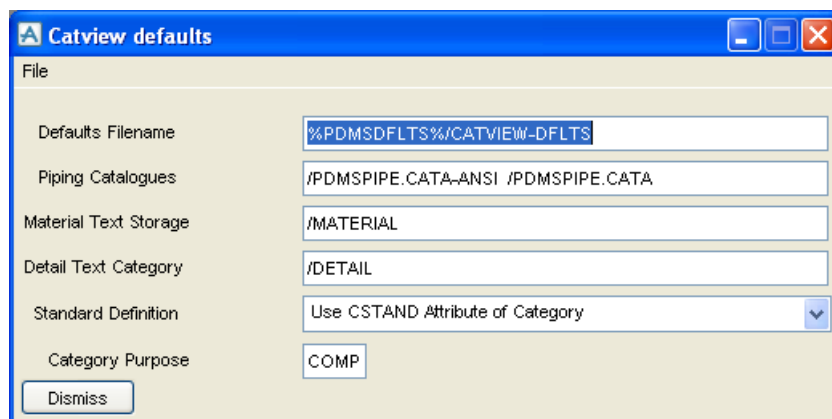


From here, all of the functions of Catview can be accessed.

## 3.1 Catview Defaults

Catview needs to be able to locate certain data within the current catalogue to enable the browsing and spec generation functions to work. The identities of what needs to be scanned in OUTFITTING are contained in a defaults file contained in your PDMSDFLTS search path.

These can be changed by accessing the **Modify>Defaults** menu.

The image shows a Windows-style dialog box titled "Catview defaults". It has a "File" menu icon at the top left. The dialog contains several input fields: "Defaults Filename" with the text "%PDMSDFLTS%/CATVIEW-DFLT.S", "Piping Catalogues" with the text "/PDMSPIPE.CATA-ANSI /PDMSPIPE.CATA", "Material Text Storage" with the text "/MATERIAL", "Detail Text Category" with the text "/DETAIL", "Standard Definition" with a dropdown menu showing "Use C STAND Attribute of Category", and "Category Purpose" with a dropdown menu showing "COMP". There are "Dismiss" and "OK" buttons at the bottom.

This form controls where to look and what to look for in the database.

### 3.1.1 Defaults Filename

This is the name of the defaults file currently loaded. It is possible to use alternative files, but the file loaded on Catview initialisation is always %PDMSDFLTS%/CATVIEW-DFLT.S. Loading and saving defaults files is done using the File menu at the top of the form.

The **Piping Catalogues** field is the name or names of catalogues to be scanned when looking for component categories. If left blank, the whole MDB will be searched.

The **Material Text Storage** field lists the name or names of items containing material descriptions. If left blank, the whole MDB will be scanned for material texts.

The **Detail Text Category** field is slightly different, because it is the name of a category or section in which new detail texts can be created if required. This must exist and the user must have write access to this area.

### 3.1.2 Standard Definition

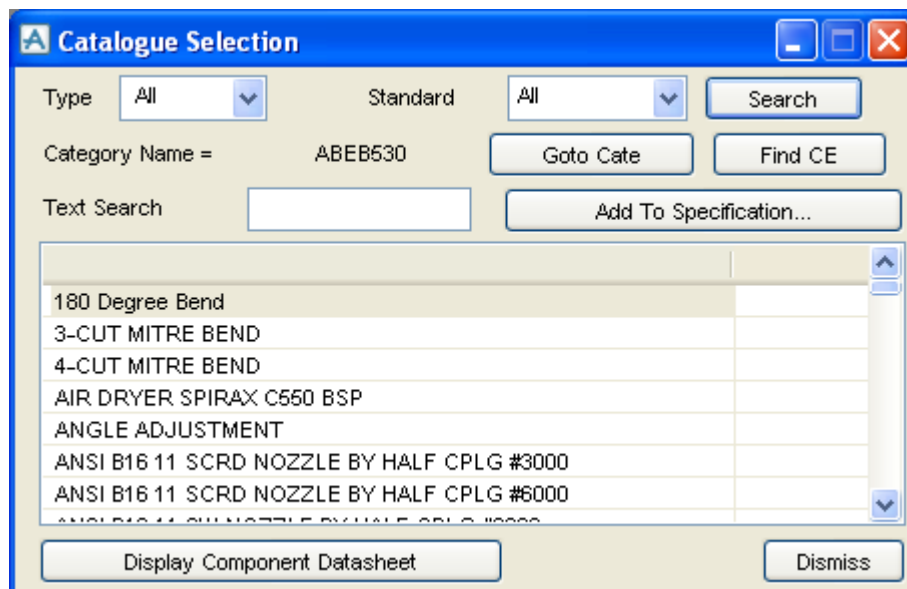
Early versions of Catview used the first character of each category name to define the category standard: A = Ansi B = British Standard D = DIN J = JIS ... New versions of Catview and the matching MAS project now use the C STAND attribute of the categories. This option should be set to use C STAND, but it is possible to select the old method if required.

### 3.1.3 Category Purpose

This is the main selection criterion for selecting piping categories. At the heart of the Catview browsing capability is the need to collect all of the correct types of category and organise the data into the correct lists. The standard Catview default is to collect all categories with the purpose attribute set to COMP.

## 4 Browsing the Catalogue

When Catview is initialised, a collection process takes place to organise the category data into a form which can be used by Catview. This is loaded into a Browser form, which enables the user to search the catalogue and navigate around the categories. This is called from the **Catview>Browse** menu on the **Catalogue Utilities** form as shown.

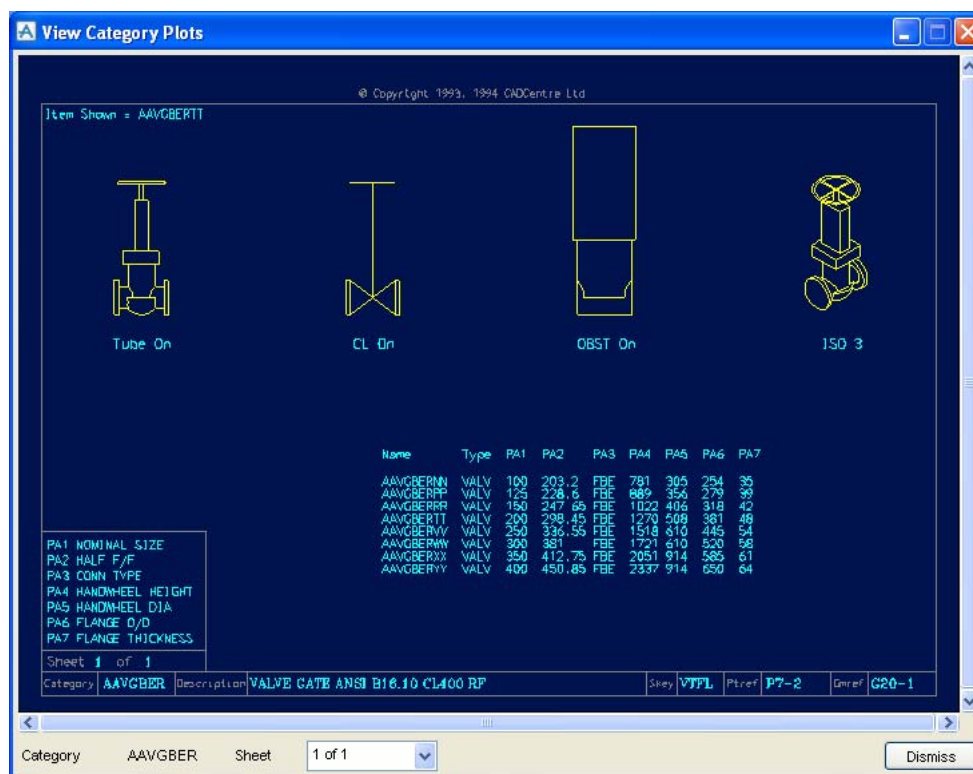


The **Catalogue Selection** browser form allows you to search the catalogues for component categories by type, standard or by text search. The form shows the default listing, which is the full catalogue regardless of type or standard. To select specific types and standards of components, the type and standard can be selected. To refresh the selection list, press the **Search** button.

A text search can be performed by typing the search text into the **Text Search** field and pressing the **enter** key. All items containing that specific text string will be included. The search is case sensitive.

## 4.1 Displaying Data Sheets

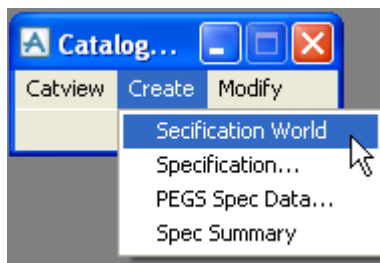
The Cadcentre standard catalogue is supplied with a series of data sheets showing the range of each category. These can be displayed by selecting the appropriate category and pressing the **Display Component Datasheet** button. A sample datasheet is shown:



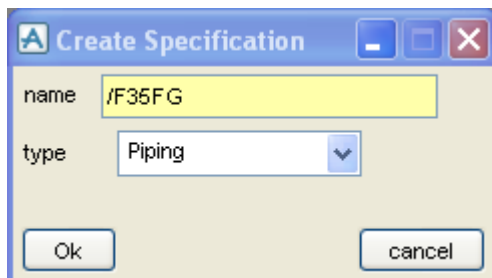
**Note:** The production of datasheets is done automatically, but the process can be difficult. The macros to produce this data are provided in the **Draftmacros** directory as part of the Catview release. See [Generating Datasheets](#) for details of how to produce Category Datasheets.

## 5 Creating Specifications

In addition to the browsing capabilities, Catview provides a mechanism for creating and editing OUTFITTING piping specifications. The menu options for this are shown:



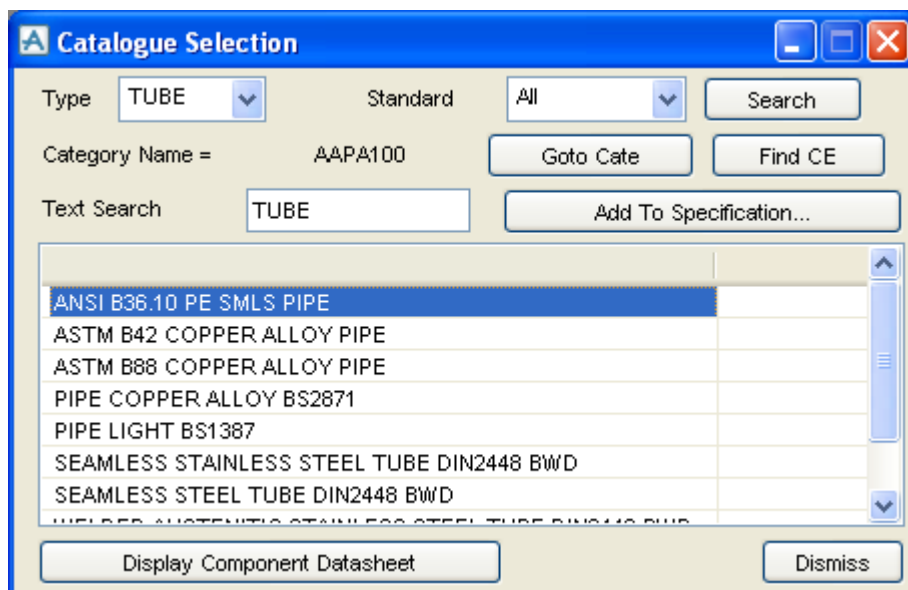
To create a Specification in PARAGON, the user needs to navigate to the required Specification World and use the **Create >Specification** menu. If no Specifications Worlds exist, these can be created first. The **Create Specification** form is shown.



The **type** option indicates the type of spec to the OUTFITTING DESIGN applications. This option sets the appropriate attributes for OUTFITTING DESIGN to find the specification.

### 5.1 Adding Components to a Specification

Catview provides a mechanism to enable a selection in the Catalogue browser to be used to create entries into a Piping Specification. This is done by selecting the appropriate category and pressing the **Add To Specification** button. A typical specification building sequence is shown:



The **Add To Specification** button lists all of the Available Specifications for the user to make a choice. In this case, F35FG is selected and the **OK** button pressed to make the selection.



This shows the Component addition form:

**Add Items To Specification /F35FG**

Category: AAPA100      Gtype: TUBE      Stype: A

**Description: ANSI B36.10 PE SMLS PIPE**

Low Size: 6      High Size: 1200

Bolting: None

**Bolting Material: Unset**

Excluded Bores      Material Text

Excluded Bores	Material Text
6	none
8	1.0038 RST 37-2
10	1.0254 ST37.0
15	1.0356.01 TTST 35 N
20	1.0356.02 TTST 35 N
25	

OK      Cancel

This form allows you to select the size range of a component, its material, stype and in the case of bolted items its bolt type and material. There is also a list of excluded sizes which removes unwanted sizes from the spec input.

When the appropriate size range and material have been selected, pressing the **OK** button will result in the selected items being built into the specification. At this point, the **Editing Specification** form is displayed, and the user has a number of options to modify and edit what is there.



## 6 Editing Specifications

The **Editing Specification** form allows the user to perform a number of editing tasks on one or all of the components in the selected Specification. The **Editing Specification** form is as shown.

Name	PBORD	SHOP	STYP	Catref	Description	Material	Compr Boltr
/F35FG/AAPA100:15 15	TRUE	A		/AAPA100DD	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:20 20	TRUE	A		/AAPA100EE	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:25 25	TRUE	A		/AAPA100FF	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:40 40	TRUE	A		/AAPA100HH	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:50 50	TRUE	A		/AAPA100JJ	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:80 80	TRUE	A		/AAPA100LL	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:100 100	TRUE	A		/AAPA100NN	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:150 150	TRUE	A		/AAPA100RR	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:200 200	TRUE	A		/AAPA100TT	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:250 250	TRUE	A		/AAPA100VV	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:300 300	TRUE	A		/AAPA100WW	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:350 350	TRUE	A		/AAPA100XX	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:400 400	TRUE	A		/AAPA100YY	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	

The editing functions of this form are as follows:

### 6.1 Changing Specification Component Names

The first option drop-down list on the Edit form deals with changing the name of Specification Components. The options all work on the selected items in the component list.

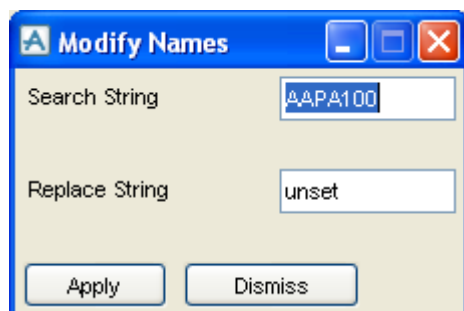
Name	PBORD	SHOP	STYP	Catref	Description	Material	Compr Boltr
/F35FG/AAPA100:15 15	TRUE	A		/AAPA100DD	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	
/F35FG/AAPA100:20 20	TRUE	A		/AAPA100EE	ANSI B36.10 PE SMLS PIPE API5L GR.B =0/0	=0/0	

There are four options for changing the Specom names by selection. These are described below:

### 6.1.1 Search and Replace

By default, Specom names are defined by [spec name]/[Category Name]:[Size 1]x[Size 2] e.g. /F35FG/AAPA100:25

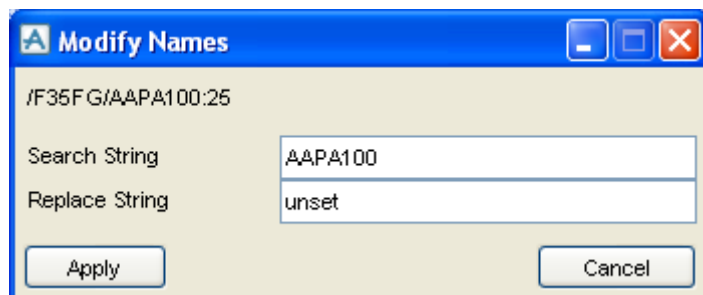
The name part - that which normally appears on an Isometric is the category name. Finally the bores sizes are added after a colon delimiter.



The **Modify Names** form presents the category name for the search and replace string as this is usually the part which needs to change. The user inputs a new name and the replace is done on all of the selected components.

### 6.1.2 Step Through

The Step Through option provides similar functionality to the first option, but allows the user to change each name in turn. To help the user, the full Specom name is shown on the form.



### 6.1.3 Removing and Adding Bore Size Descriptions

When Catview creates components, it looks at the number of connection points which potentially have different bores. If the pointset contains more than one bore parameter, it assumes that the component has more than one bore and creates the spec headings accordingly. Some items such as blind flanges have two bores in the catalogue, but they are the same value. If these are placed in a spec with other flanges, there is a potential mismatch of spec headings. Conversely, adding reducing flanges to a set of parallel flanges will also cause a mismatch.

The removal or addition of bores works in two operations:

- Changing the name
- Changing the selectors

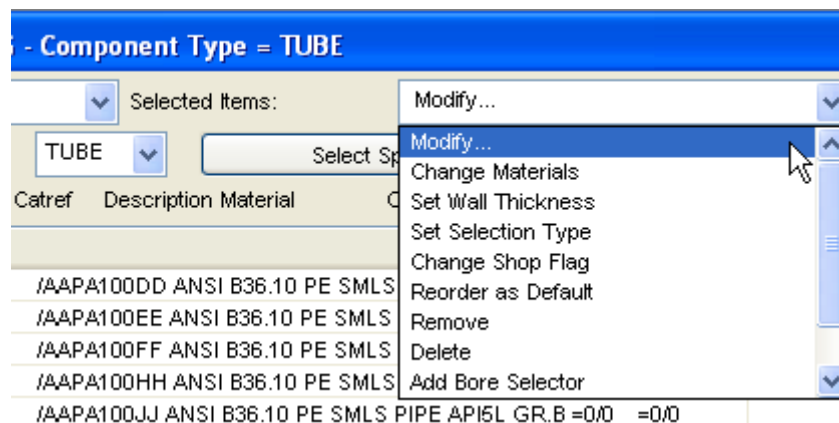
The rename option under naming simply adds or removes the extension which contains the bore sizes.

i.e. **/F35FG/AAPA100:25** could change to **/F35FG/AAPA100:25x15**.

Changing the selectors is done under the Modify Option and is described later.

## 6.2 General Modifications to Selected Items

Apart from changing the name, there are a number of options which can be applied to the selected list. The general modification menu is shown:



The “Modify” option shows the **Edit Specification Component** form, a general Specification editor form for each component. This is as shown:

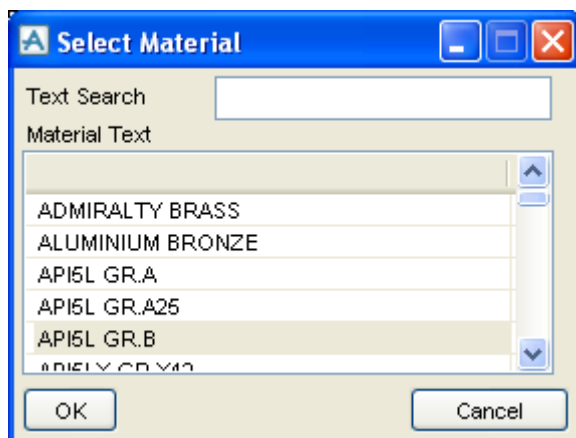
The screenshot shows the 'Edit Specification Component' form. It has a title bar with the text 'Edit Specification Component'. The form contains several fields and buttons. The 'Name:' field is set to '/F35FG/AAPA100:50'. The 'current bore = 50' is displayed next to it. The 'Stype:' field is set to 'A'. There are 'CE' buttons next to 'Catalogue Reference:', 'Detail Reference:', 'Material Reference:', 'Bolt Reference:', and 'Component Reference:'. The 'Catalogue Reference:' field is set to '/AAPA100JJ'. The 'Detail Reference:' field is set to '/AAPA100-D'. The 'Material Reference:' field is set to '/ALB'. The 'Bolt Reference:' field is set to '=0/0'. The 'Component Reference:' field is set to '=0/0'. There are 'Goto Component', 'Goto Detail', 'Goto Material', 'Goto Bolt', and 'Goto Cmpref' buttons. There are also 'Apply', 'Reset', 'Skip', and 'Cancel' buttons at the bottom.

This form provides facilities to change all of the Speccom attributes and to navigate to different parts of the database using the **Goto** buttons. The **CE** buttons are used to set the

relevant attributes to use the current element in the OUTFITTING database. A check is made to ensure that the element is of the correct type.

## 6.3 Changing Component Materials

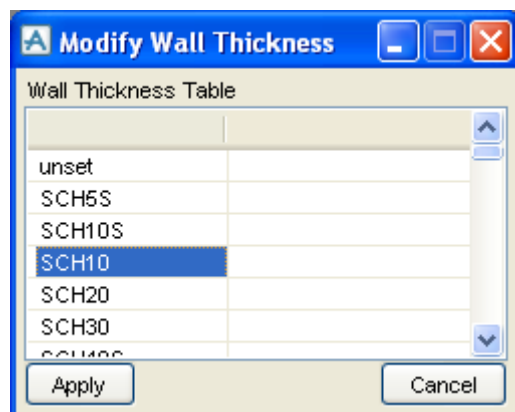
Selecting the **Change Materials** option displays the **Select Material** form.



From this, the user selects a new material and presses the **OK** button to change all of the selected components. The **Text Search** enables materials to be selected by searching for a text string.

## 6.4 Modifying Wall Thickness

Setting the wall thickness in Catview is done by appending the wall thickness text to the standard detailing text and creating a new SDTE element. The new SDTE element is created in the same directory as the standard detail text (usually the component category). If this cannot be accessed, the new detail is created in the defaults area.



## 6.5 Setting the STYP

By default, the type for all components is A. This can be changed at the point when the components are added to the spec or it can be changed using the **Reset Style** form shown:



## 6.6 Changing the Shop Flag

The Shop Flag is used to differentiate between fabrication and erection items on piping isometrics. Shop TRUE means that the item is to be fabricated in the workshop and shop FALSE means it should be located at the site. The shop values are set by default for certain items, but this can be overridden by the **Change Shop Flag** option. There is no form for this action, the flag is simply swapped to its opposite value.

## 6.7 Setting the Default Selection Item

The order of components at each particular size determines how the list of components are displayed on the **CHOOSE** form in OUTFITTING DESIGN. It also controls the default item when the select command is used. To change the selection list, the required items need to be selected on the form and the **Reorder as Default** option selected.

### 6.7.1 Removing a Specom from the Specification

The action of removing a Specom takes the item from the spec and places it in the nearest available limbospec. This is done, by selecting the items to be removed, and selecting the **Remove** option. If a limbospec does not exist, one is created.

## 6.8 Deleting a Specom

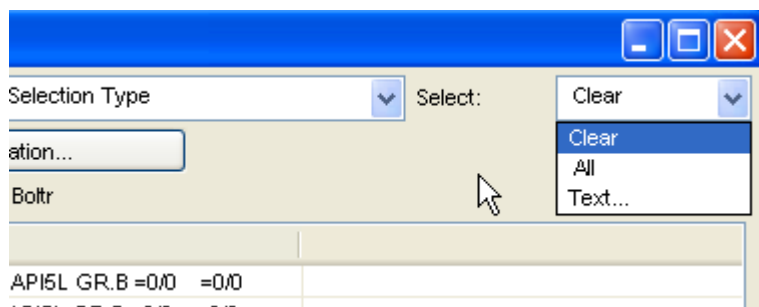
Deleting a Specom deletes the element entirely and all references to it will no longer work. This can be done by selecting the required components and using the **Delete** option.

### 6.8.1 Adding and Removing Bore Selectors

Operations such as adding reducing flanges to a range of parallel flanges will result in a mismatch of headings. Some items may have two bore sizes and some only one. To rectify this, there are options for adding or removing a bore selector. To do this, select the items to be changed and the appropriate action. Currently, Catview only deals with 1 or 2 different bore sizes.

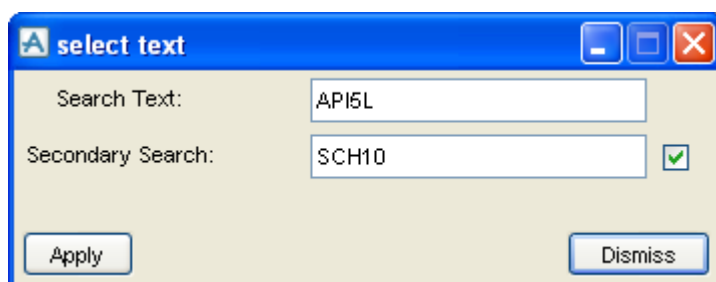
## 6.9 Selecting Items from the Component List

The selection options provide a quick method of selecting like items. This is useful when dealing with large numbers of components. The selection options are shown below:



The **Clear** option clears all selections.

The **All** option selects all items in the list. The **Text** option allows single or double level selection by matching text strings in the list. The text selection form is shown:



## **7 Other Catview Options**

### **7.1 Create PEGS Spec Data**

This option produces a macro for mapping the spec details with the PEGS system. The user is prompted for an output file, which needs to be run into PEGS as a macro.

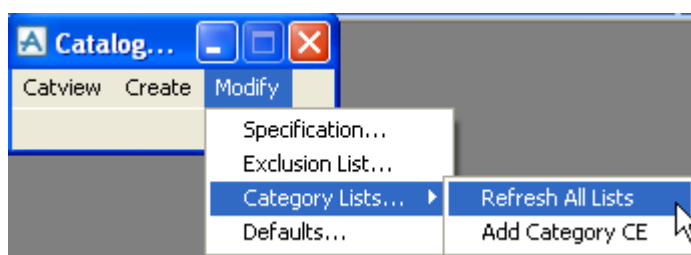
### **7.2 Create Spec Summary**

This outputs a spec summary to file. The user selects a spec and a file name for the output.



## 8 Catview Modify Options

The Full Modify menu and their functions are as shown:

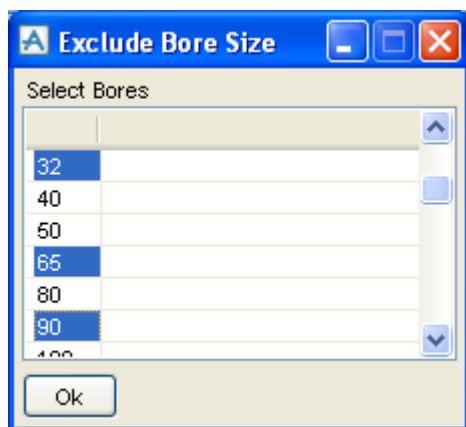


### 8.1 Modify Specification

This allows you to select a specification and brings up the main Component Editor form.

### 8.2 Modify Exclusion List

Bore sizes like 32mm 90mm 125mm and 175mm are normally not used, although we have some components at those sizes. The Exclusion list enables these non-preferred sizes to be ignored by the specification building process. This option shows the **Exclude Bore Size** form, which maintains the default list:





## 9 Category Lists

The main browsing functions of Catview rely on data gathered when Catview starts. These are lists of category names, types, descriptions and standards that are stored under the browsing form. This data is collected from attributes of the component categories and forms the backbone of the system. The attributes, which are used, are as follows:

<b>Purpose</b>	The databases is scanned for categories with the appropriate purpose. The default purpose is <b>COMP</b>
<b>Description</b>	The description attribute provides the text description of the category contents. This is used by the browser form for component selection.
<b>Cdetail</b>	This is a reference attribute which points to the default detail text for the category. Each component, which is added to a specification, will reference this standard text.
<b>Gtype</b>	This is a word attribute and it is used to sort the categories by generic type such as TEE, FLAN, ELBO etc.
<b>Cstandard</b>	This is a text field which is used to indicate the category standard, such as Ansi, Din or BS. "none" is also used.

During a Catview session, these lists are not normally modified, but it is possible to refresh the lists to reflect new categories or other modifications to the catalogue. This is done using the **Modify>Category Lists>Refresh All Lists** menu.

Alternatively, to add a new category without regenerating the full lists use **Modify>Category Lists>Add Category CE** when positioned at the required category or below.



## 10 Bolting

One of the main aims of Catview has been to make the system independent of names and naming conventions. The only thing, which does rely on names, is bolting.

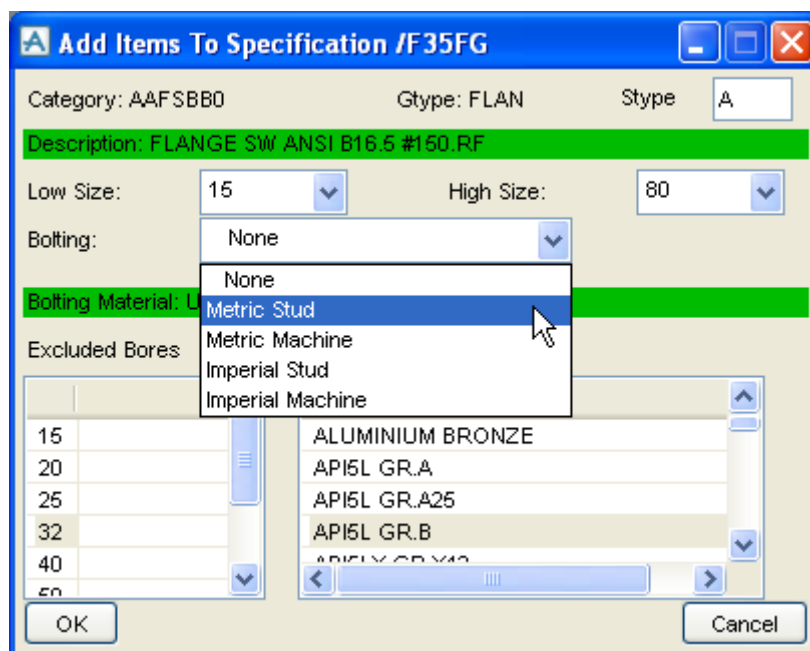
This is based on the Cadcentre component naming conventions, which are used in the MAS project catalogue. The system uses the 5<sup>th</sup>, 6<sup>th</sup> and 8<sup>th</sup> characters to map onto a bolt with a similar name.

e.g. flange component /AAFSBB0JJ will map onto BBJ for its rating facing and size. The bolt set for this is could be an imperial stud bolt and this would use the characters IS, so the bolt for this component would be /ISBBJ.

**Note:** Because of this mapping, the bolting side of Catview may be inappropriate for those companies not using the Cadcentre Catalogue. This does not however remove the benefits of Catview for general specification work and Catalogue browsing.

Adding bolts to a component set, needs to be done during the specification building process. Selecting one of the four bolt types from the list such as “Metric Stud” will show a bolting materials selection form and then proceed to add the bolts to the specification at the same time as it adds the components.

Missing items will generate an error message and be ignored.





## A Generating Datasheets

The files in Catview/draftmacros directory are designed to be able to produce automatic catalogue documentation in the form of OUTFITTING DRAFT sheets for plotting or screen viewing. The directory contains the following files:

SETUP.MAC	The macro for building an MDB and all of the necessary elements for the process to work
draftmac	The main macro for building the documentation sheets
cates	An example file containing a list of categories to be documented
CATE-PARA	A file showing the current values of the category OUTFITTING DESIGN parameters for the Cadcentre Catalogue
catepara.mac	This is a macro to read the file CATE-PARA and set the required uda values for the Cadcentre Catalogue.
plot1, plot2	2 example PLOT files

### A.1 How It Works

The basic idea of the draftmac macro is as follows:

1. Input the name of a category to be detailed (from the file called cates)
2. Go to the category and read the parameter texts
3. Visit each component in the category and compile a list of components and their parameters
4. Calculate how many sheets are required and split the data accordingly
5. Create new sheets and write the text info
6. If the component has no Geomset, the text area is adjusted to fill the sheet.
7. Assuming that the component has a Geomset then draw 4 views of a component showing the different representation types (tube on, CI on obst on and an iso view)
8. Read the next category name and repeat the process

### A.2 Getting Started

Drawing catalogue components in DRAFT is not quite as straightforward as it would first appear. In fact we cannot draw catalogue components directly so we have to resort to drawing OUTFITTING DESIGN items instead. This is done by having a single piping

component in OUTFITTING DESIGN, and swapping the catref of its Specom, for each category change. To do this, we need to be able to write to the catalogue from DRAFT and we need to visit OUTFITTING DESIGN each time to update the spatial map. The macro SETUP.MAC is designed to build all of the data required for this purpose and to change module access. Before you can run this, you need to change line 10 to include the name of your master catalogue database.

After this, the macro should be run from MONITOR

**Note:** Because the drawing macro needs to run in and out of OUTFITTING DRAFT, it is important to be able to get in and out of OUTFITTING DRAFT without appware errors. This requires the addition of the sample data from the paddle DB. This can be loaded via the ADMIN Load Sample Data menu.

Finally, you need to go into ADMIN to change fontfamily 1 to be proportionally spaced. This is so that the columns of parameters come out nicely aligned. To do this, enter ADMIN and type - FONTF 1 IR 6 STYLE 7.

After the set-up is completed, you need to create a file called cates, which contains the name of all of the categories you want to document. Say this is called c:\temp\cates, you can go to OUTFITTING DRAFT and start the documentation process with the command:

```
$M/c:\cadcentre\Catview11.3\draftmacros\draftmac c:\temp\cates
```

## A.3 Catalogue Requirements

The first part of the process for each sheet is to go and find the parameter description text. This is assumed to be in two forms:

1. An ADTE element with the category description as its name e.g., / ANSI.B16.5.BW.ELBOW
2. A series of text elements containing the parameter descriptions in the form /AABJ0JH-PA1 with an stext which describes the purpose of each parameter e.g. STEXT 'NOMINAL SIZE'
3. Each category is assumed to have an equivalent DTEX element with a name in the form /AABJ0JH-D i.e. category name plus '-D' This DTEX element allows us to use the description text for the component types and also allows us to find the appropriate SKEY
4. Each category now contains a set of UDA's which allow you to set certain drawing parameters (Like model set in PARAGON). These include:

:VIEWPAR	(View Direction. Default N)
:DDHEIGHT	(DDHEI for nozzles etc [n * param 1])
:DDRADIUS	(DDRAD for bends etc [n * param 1])
:DDANGLE	(DDANGLE Default 90)
:VSCALE	(Scale modifier if all else fails Default = 1)
:VNOTE	(Special note to be put onto the drawing - Default = unset)
:DLOG	(logical attribute to control if a drawing is produced - Default = True)

## A.4 Setting Up Catalogue UDAS

If you leave the above UDA settings at their defaults, the draftmacro should still work, but you might find that for some elements, the view direction or OUTFITTING DESIGN parameters need to be set to get better results.



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