

CATIA DELPHI Standard

Supplier Use

CATIA DRAWING STANDARDS

The following standards are to be respected by CATIA designers and detailers to avoid discrepancies from DELPHI THERMAL practice.

WORKING PRACTICES

Within each new project, there exists a start model (corresponding to the standard required, customer or DELPHI). When commencing a new component this model **must** be used and then re-filed in the designers own work file.

(see *APPENDIX B - CATIA MODEL NAMING CONVENTION*)

The purpose of using a start model is that all the following model standards are set. i.e. colours, filters etc.

When making a change to a production part the latest version will normally be found in the relevant archive (ARCHIVE) file. However it should be verified that a change is not currently being made. Check with your Lead Designer to locate the latest level to work on.

SETS

One set only for each part including changes; PART NUM XXXXXXXXX

LAYERS FOR MODIFICATIONS

A release may be the first production release, a release of a single change or a release of a package of changes.

For the first release, all elements are to be in layers 1-99.

For subsequent releases all new/modified elements to be in the usual layer + 100 and in addition each change within the release to be identified by applying different graphic colours as described in the table below.

Therefore, on acquiring a model for change, initially all elements to be returned to layers 1-99 and any graphic colour removed.

N.B. For the first change in a release the space elements do not require any graphic colour since they will take the layer colour yellow. However, graphic colour yellow must be applied to the corresponding draw elements.

Where the release level is raised due to the Engineering Release of a part number, the modification column colour for the release should be the same as the release authority change.

CHANGE	SPACE ELEMENTS	DRAW ELEMENTS
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No.				
1	No graphic colour	(yellow by layer)	Graphic colour 5	(yellow)
2	Graphic colour 95	(pink)	Graphic colour 95	(pink)
3	Graphic colour 110	(cyan)	Graphic colour 110	(cyan)
4	Graphic colour 75	(mustard yellow)	Graphic colour 75	(mustard yellow)
5	Graphic colour 60	(medium blue)	Graphic colour 60	(medium blue)

These attributes could be generated automatically by choosing the following options on AUXVIEW2 Function:

DEFAULT

DRAW GENERATED GRAPHICAL ATTRIBUTES set to SAME

DIMENSIONS

ANSI, DSUS Standards

Text heights/widths as per start model.

LINETYPES

Use only standard CATIA linetypes 1 to 6 as below:

1 SOLID	Construction lines (space mode)
2 DOTTED	Hidden features (draw mode)
3 DASHED	Section lines (draw mode)
4 DOT-DASHED	Centre lines (space and draw mode)
5 PHANTOM	Clarification of features (draw mode)
6 BREAK	Partial sections and views (draw mode)

PATTERNS

These have been set up as per CADAM Areafill with corresponding description names.

AUXILIARY VIEWS

The Lead Designer decides in his group which method should be used to create the auxiliary views (Graphic-Dress Up-No Show, Drawspace Project or Auxview2)

Third Angle Projection, to Harrison / GM draughting standards.

PLAN VIEW

SIDE VIEW

FRONT VIEW

PICTORIAL VIEW

FRAME VIEW

The above views are contained in the "Harrison start model" and may be modified or deleted as required. All other auxiliary views to be created as required in DRAW mode.

VIEW NAMES

In naming both sections and views use letters once only, do not use letters I, O or Q. On completing alphabet use two different letters. Convention for sections and views as follows:

SECT A-A
SECT B-B
VIEW C
VIEW D
SECT AB-AB

Doubles ie: AA-BB or combinations preceding the doubles ie: BA-CB are not allowed.

DETAILS

Certain details have been included in the Drawing Start Model for some standard notes, DITTO as required. If a detail is to be modified with several different copies, it may be exploded.

GENERAL: DITTOS/DETAILS **used** must not be exploded
DITTOS/DETAILS **unused** must be deleted

LIBRARY DETAILS

Master and Ghost Formats have been set up in the library "LIBRARY", family "DELPHI-E FRAME" or "DELPHI-H FRAME". Choose correct size and always ditto format onto FRAME VIEW auxiliary view and edit details. **DO NOT DELETE**. If using the roll format read "DELPHI-H RO LEFT, DELPHI-H RO MIDD, DELPHI-H RO END" details (these are not available in "DELPHI-E" family). The "GHOST FRAME" indicates the no-draw zones as demanded by Harrison standards.

DRAFTS

One draft only to be used.

TEXT: accessible with STARTUP Function in TEXTD2, MANAGE

ANSI, DSUS Standards

DESCR1: H=4.16mm W=3.05mm Char space=1.40mm Line space=2.08mm -General notes

DESCR2: H=6.25mm W=4.58mm Char space=1.40mm Line space=3.12mm -Drawing numbers

DESCR3: H=2.30mm W=1.83mm Char space=1.40mm Line space=1.15mm -Tolerance notes

DESCR4: H=3.00mm W=2.20mm Char space=1.40mm Line space=1.50mm -Revision column

ASSEMBLIES

Each assembly part to be in its own set.

COMPONENT ORIENTATION

Details where possible to be drawn about their own axis.

WINDOWS

PLANXY
SIDEXZ
ENDYZ
PICTORIAL
DRAW

SCREENS

SPC/DRAW SPLIT
STD4 WINDOW

N.B. These stored windows and screens will need modifying to view actual model detail.

SURFACE REQUIREMENTS

All fillet and edge radii above minimum cutter radius to be surfaced.

All changing radii to be surfaced.

Face, solid and volume requirements to be determined by Lead Designer.


N.B. IF IN DOUBT, ASK YOUR LEAD DESIGNER.

LAYER CONVENTION

Layers	Elements	Colours																			
0	No elements																				
1	Wireframe inner	DARK GREEN	(47)																		
2	Wireframe outer	GREEN	(3)																		
3	Surface inner	DARK BLUE	(122)																		
4	Surface outer	BLUE	(4)																		
5	Split lines	WHITE	(1)																		
6	Slide direction line	WHITE	(1)																		
10	Volumes and Skins	WHITE	(1)																		
11	Faces inner	DARK RED	(32)																		
12	Faces outer	RED	(2)																		
13	Solide	GREEN	(3)																		
15	Profile of a line	GREEN	(3)																		
21	Section A-A (space elements)	WHITE	(1)																		
22	Section B-B (space elements)	WHITE	(1)																		
23	Section C-C (space elements)	WHITE	(1)																		
24	Section D-D (space elements)	WHITE	(1)																		
25	Section E-E (space elements)	WHITE	(1)																		
89	Section ??-?? (space elements)	WHITE	(1)																		
90	Plane	CYAN	(110)																		
91	Solide Wireframe	GREEN	(3)																		
101	Modified Wireframe inner	<div>Colour dependant on change number</div> <table><thead><tr><th>CHANGE N°</th><th>COLOUR</th><th>SPACE ELEMENT COLOUR</th></tr></thead><tbody><tr><td>1</td><td>5</td><td>yellow by layer colour</td></tr><tr><td>2</td><td>95</td><td>(pink)</td></tr><tr><td>3</td><td>110</td><td>(cyan)</td></tr><tr><td>4</td><td>75</td><td>(mustard yellow)</td></tr><tr><td>5</td><td>60</td><td>(medium blue)</td></tr></tbody></table>		CHANGE N°	COLOUR	SPACE ELEMENT COLOUR	1	5	yellow by layer colour	2	95	(pink)	3	110	(cyan)	4	75	(mustard yellow)	5	60	(medium blue)
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124	Section D-D (space elements)																				
125	Section E-E (space elements)																				
199	Section ??-?? (space elements)																				
200	Frame (draw mode)	WHITE	(1)																		
201	Text all sizes and all views	WHITE	(1)																		
202	Dimensions all types, sizes and associated text	WHITE	(1)																		
203	Patterns and shapes all types	BLUE	(4)																		
204	All draw data	WHITE	(1)																		
210	Grids, balloons & associated text	WHITE	(1)																		
245	Space and draw axes only	WHITE	(1)																		

CATIA Model Dimensions

Each customer has its own standard regarding the Catia model dimension. This criterion defines the precision of some elements such as intersection of curves to define a point, or the distance between two curves to be considered as identical.

GEOMETRIC STANDARDS		
	MODIFYING THESE VALUES MAY	
	LEAD TO UNPREDICTABLE RESULTS	
TOLERANCE	CURRENT	RECOMMENDED
IDENTICAL CURVES	.0200000	.0200000
INTERSECTION PROJECTION	.0002000	.0002000
INFINITY	20000.00	20000.00
BENDING	.0060000	.0060000
STEP	4.000000	4.000000
MODEL DIMENSION	2000.000	

Value to change regarding the customer standard

To obtain this page, please take the following procedure:

Function: *STANDARD*
Function: *MODEL*
Select *PAGE 2*
Select current value for *'MODEL DIMENSION'* and type the appropriate value
ENTER

When you change the 'MODEL DIMENSION' value, all the other values are automatically implemented to keep a good concordance.

The best way to avoid some CATIA model dimensions mistake is to use the customer Start-up model. With this, you are sure to be in complete accordance with your customer standard, regarding all aspect of it (model dimensions, graphic attributes, filter and layer convention,...).

The current model dimensions imposed by DELPHI customers are:

CUSTOMER	MODEL DIMENSION
AUDI	2,000
BMW	5,000
DAEWOO	10,000
DAIMLER CHRYSLER	2,000
FIAT	10,000
PSA	10,000
PORCHE	2,000
RENAULT	5,000
SAAB	10,000
VOLKSWAGEN	2,000
DELPHI	2,000

NB:

Model dimension value must be changed when the model is initially created!

Later modification would not have the required impact!

CATIA FILE PROCESS

When making a change to a production part the latest version will normally be found in the relevant archive (ARCHIVE) file. However it should be verified that a change is not currently being made. Check with your Lead Designer to locate the latest level to work on.

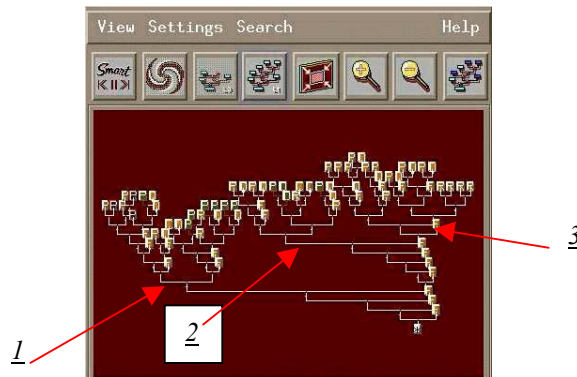
1. The modification is made to the relevant model, and is re-filed in the designers (WIP/xxxxxx) file. The designers initials are placed in location 46-48, the relevant issue number in location 52-53, and a new date in location 65-70.
2. On completion, the designer writes the letter “C” in location 43 and informs his Lead Designer that the job is ready to be checked.
3. The Lead Designer copies the file in his directory and checks it. If corrections are required, he tells the designer to do them. After re-work, a new date must be given to the model, location 65-70.
4. Once satisfied, the Lead Designer moves the model to the checked (CHECK) file, changes the designers initials to “LUX” and writes the letter “F” in location 43. He creates a plot, collects the signatures and tells the designer to delete the model in his directory.
5. Once the signatures are collected, the Lead Designer adds the signatures in the change column and fills out the Drawing Transfer Request Book.
6. The CATIA supervisor transfers the identified drawings from the checked (CHECK) file to the archive (ARCHIVE) file. Once completed, the previous drawing release is transferred from the archive (ARCHIVE) file to the history (HISTORY) file.
7. Once the transfer to archive (ARCHIVE) file is done, the Lead Designer gives the package to administration for distribution.

CATIA SOLID GENERATION

The following points are **suggestions of working practices** to be followed in model construction prior to creating solids. Some items refer to wireframe construction, some to surfacing/facing and some to the actual solid creation.

- Use a start model corresponding to the customer standard (this first point is not a suggestion but a necessity to avoid some future problem when the file will be transferred to the customer design department).
- Wireframe should always be created by intersecting *planes* with *planes* or *surfaces*, use *planes* in preference to planar *surfaces* (the *CATIA.Dynamic.Sketcher* could be used too).
- All elements must be in the same set. This is a requirement to have the elements logically linked, although this contravenes the CATIA DRAWING STANDARDS.
- The best way to create a solid is to have a *features tree* (visible on the *part editor*) as flat as possible. The primitives must be grouped by any criteria which can clarified the construction (ribs with ribs, holes with holes, ..., or by geographical position on the main body):

Good features tree:
3 pre-assembly before the general union



Bad features tree:
Fillet features are generated step by step instead of grouped on the same primitive).

In general, keep the model as simple and clean as possible, by adopting a modelling strategy based on easy-to-understand, editable features:

Ref.: Online *CATIA Documentation SOLIDE* page 32

- Keep the branches of the Part editor logical:
 - * Do not add other solids to the same type of solid/features on a *branch*
E.g.: a branch having 3 pre-assembly snaps should not have one boss added to it before the *union* with the main body, but the entire *branch* having all bosses on it can be united with this *branch* before the final *union* with the main body.
 - * Try to build entire areas in one *branch*. Use the *reorder* feature to re-arrange the model structure.
- The primitive modelling is really important and must be done with many attentions; in fact the drawing generation will be easier if the part is built with some simple primitives. For instance, it is better if an axes is created by *cylinder* feature and not with *revolution* or *prism* feature, the drawing dimension will be generated directly in diameter dimension and not in distance dimension.
- It is good practice to generate *fillet* or *draft* feature directly on the primitive and not at the end of the construction. In the same way, in case of removal primitives to the part, all operation as *fillet* or *draft* must be performed before the operation on the main body (*subtraction*, *union*, *intersection*,).
Keep the fillet logical
 - * Do not add edges from different areas to the same *fillet* feature.
- In case of primitive whose must be *split* used a *plane* instead of another primitive, this will reduced the number of primitives in the model). In the same way, the *split* operation is possible by using the part (pick a face of the *solid* to define the *split* element).
- In case of modelling with some relative *axis*, the absolute one (**axis1*: corresponding to the vehicle reference) must be activated before the model is sent to the Lead Designer (according to the CATIA DRAWING STANDARDS).
- For plastics cases **ONLY**:
 - * Do not create *fillets* smaller than R0.8 unless required by the geometry: E.g.: inside thick walls.
 - * On all other parts (levers, snaps,), **all the blends should be modelled.**
 - * Cases will have constant thickness, unless otherwise required by the customer. Therefore, it is generally preferred to model the inside and perform a *shell* feature for generating the part walls.
- Elements used for sweeping profiles should be smooth, do not used *spline* with a lots of *arcs* unless required by customer.
- *Prisms* are preferred to *sweeps*, so use them wherever possible.
- Perform *Force Update* and */cln* at least once every 8 hours, and before sending the model.
- Send the files *unsmart* to reduce *Catexp* size.

Ref.: Online *CATIA Documentation SOLIDE* page 36.

Assemblies and Assembly Drawing

A) ASSEMBLIES

- 1 All assemblies will be named using the naming convention in Appendix B of the “*Catia users information*” guide.
- 2 Create the assembly using the VCS (Vehicle Coordinate System) as the active coordinate system.
- 3 We cannot deliver any full assemblies in Catia. Since these are being created using *sessions*, and *sessions* cannot be exported or sent to the customer. However, this does not mean we do not need to create a *session*. The *session* is a mandatory to be created for assembly validation purposes, as well as for drawing purposes.
Although not a deliverable, creating the assembly *session* will be the first step for relating the deliverables for the assembly.
- 4 Part that are modeled in a 3D location than the one required in a particular assembly (i.e. parts common for several assemblies/vehicle environments) are to be repositioned at the session level using the following features:

MODEL
MODIFY
DUPLICATE
MOVE
MULTISEL (separate screen)

After that:

FILE
CLOSE (the part).

The next time you open the *session*, the part should be in right place.

- 5 For each part having multiple uses in the assembly, we need to build a separate *model* file, where that particular part will be defined as *ditto* and instanced in all the locations required by the assembly. All these *model* files are deliverables to the customer, as part of the assembly file.
- 6 CHECK for INTERFERENCES and MISMATCHES.

B) ASSEMBLY DRAWING

- 1 All assembly drawings will be named using the naming convention in Appendix B of the “*Catia users information*” guide.
- 2 The assembly drawing is a separate model file, containing only the drawing.
- 3 For creating the assembly drawing, please follow the steps as described:
 - Open the assembly *session*, which in turn will call all the components needed in the assembly.
 - Open a new *model* file, which will be added to the *session*, in which you will create the assembly drawing. Do not add any *models* to this new file.
 - Do not save the *session*, as this will lead to the opening of the assembly drawing every time the assembly *session* is opened.

Every time you need to edit this file, you first will need to open the *session* in order to have the full assembly opened.

- 4 Use only *AUXVIEW2* for creating the drawing.
- 5 Keep in mind, that if a part is modified and the session is opened with the modified part, you will need to *update* the drawing view. But in case of a dimension affected by the modification, **the drawing entity and the dimension concerned will stay in place**, and so you will be obliged to delete them, and to recreate the good dimension attached to the good drawing element.

APPENDIX B

CATIA MODEL NAMING CONVENTION

01.....08	10.....42	43	44	46..48	49..50	52..53	55.....63	65...70
HR123456	CASE UPPER SURFACES	U	A	RJR	01	01	76301-01A	01JA00

01 to 08	=	PART NUMBER (REF PROCEDURE)
10 to 42	=	PART DESCRIPTION (GM STANDARD)
43	=	STATUS CODE *
44	=	MODEL TYPE *
46 to 48	=	DESIGNER / DRAUGHTMANS INITIALS
49 to 50	=	SHEET NUMBER
52 to 53	=	ISSUE NUMBER
55 to 63	=	PROJECT NUMBER IDENTIFICATION (REF PROCEDURE)
65 to 70	=	DATE (DDMMYY) *

* STATUS CODE:

- U = UNFINISHED
- C = TO BE CHECKED
- R = REFERENCE DRAWING ONLY
- P = PROTOTYPE RELEASE
- F = FINISHED (PRODUCTION)

* MODEL TYPE:

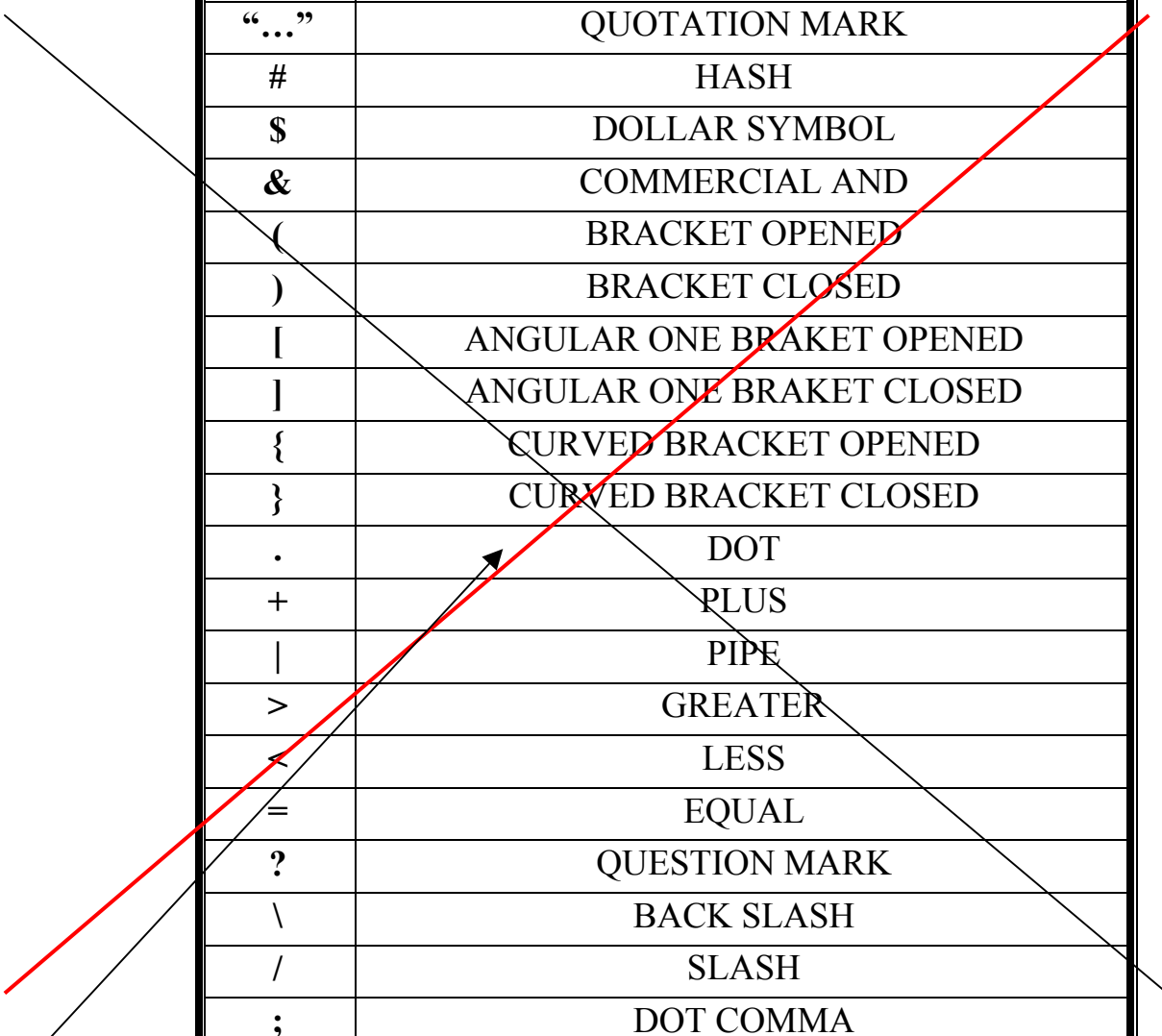
- A = SPACE MODEL ONLY
- B = DETAIL / MODEL (SPC & DRW)
- D = DETAIL ONLY

* DATE MONTHS: JA / FE / MR / AP / MY / JN / JL / AU / SE / OC / NO / DE

NB: The best practice is to use ‘_’ (Underscore) instead of ‘space’, in fact all Unix Platforms do not recognise ‘Space’ as the same character and it could generate some exchange problems.

CHARACTERS CONVENTION:

The following characters are not allowed in the model name:



'	INVERTED COMMA TO LEFT
`	INVERTED COMMA TO RIGHT
,	COMMA
!	EXCLAMATION MARK
"..."	QUOTATION MARK
#	HASH
\$	DOLLAR SYMBOL
&	COMMERCIAL AND
(BRACKET OPENED
)	BRACKET CLOSED
[ANGULAR ONE BRAKET OPENED
]	ANGULAR ONE BRAKET CLOSED
{	CURVED BRACKET OPENED
}	CURVED BRACKET CLOSED
.	DOT
+	PLUS
	PIPE
>	GREATER
<	LESS
=	EQUAL
?	QUESTION MARK
\	BACK SLASH
/	SLASH
;	DOT COMMA
°	DEGREE SYMBOL
±	PLUS OR MINUS SIGN
␣	TABULATION