

CATIA 快捷键-方法 (CATIA shortcut - method)

CATIA shortcut set some of the tips just in CATIA shortcut set, is originally wanted to call a quicker way to set up, look at some of the information, as if you were nothing special, just tell me about my CATIA shortcut set.

Into you want to setup shortcut module, at the command of the margin right-clicking choose custom, and then in the pop-up window, select the COMMOND, create a new command, the catia interface of a blank on command. Click the command window, find the command you want to, drag to the blank command bar, if you don't do some commands in a single module is set no shortcuts, need to convert to the appropriate module to set, very troublesome, in accordance with the above method will not. Once you have to drag the command from the shortcut key to the blank command line, all commands can set the shortcut key. I usually set it to (ALT + letter), my thumb pressed on ALT, and other fingers to find commands. If the module is converted, use CTRL + letter.

I don't know if CATIA can menu shortcuts like UG set letters, because it seemed to be just the input letters, CATIA will default to the macro command input, not as a shortcut to identify, very troublesome, hope to have directly!

Recently, with the guidance of gao people, I finally know how to set CATIA shortcut keys with single letters.

Into CATIA options, in the general options, there is a

option, let you choose (P1, P2, P3 and p points, and then turn off the CATIA, open again, you can set shortcuts for individual letters, don't need to use the key combination.

A little bit of personal care, hope for everyone! My email address is henryyan308@126.com and I have the opportunity to communicate with you

If you want to select all of them, you can do it with Ctrl + F, you can do it in the name input *, but you don't have catia in hand, you haven't tried

How to select all sizes:

Each label has its own name, Dimension. X (1, 2, 3...). Just type Dimension in the name column of search. * will do

CATIA keyboard shortcut highlights

F3 -- - hide the directory tree

F1 -- - real-time help

CTRL + z -- -- -- -- -- -- -- -- to withdraw

CTRL + y -- -- -- -- -- -- -- -- redo

CTRL + n -- -- -- -- -- -- -- -- new

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CTRL + mouse button view zoom in and out

Alt + mouse key view translation

Alt + mouse button + mouse button view rotation

Alt + Enter = property

Shift the key in the red block and drag

You can zoom in and out quickly and it works very well

Press CTRL and then the middle key is zoomed out

I'm just going to hit the middle button and CTRL is the object rotation

When the object is rotated, a red circle appears outside

In the circle is the rotation of the XYZ axis

And outside the circular region is a particular rotation of the Z axis

Pointing to a closed hollow surface with a mouse,

Then press the arrow keys, and you can select the inner surface!

CTRL + F11, show object selector.

Ctrl + Page up - zoom in

Ctrl + Page down

Alt + shift + up and down arrow - rotate

Ctrl + up and down arrow -- pan

Ctrl + shift + left and right arrows -- -- rotate

Shift + F3 -- work on specification tree

Shift + F2 -- specification tree overview (in tool bar).

Can be customized

In free-form and a-level surfaces:

F5 -- the "operation plane dialog box" (especially important for "n-dot" commands)

CATIA13 asked

1. How to quickly know the proportion of the parts in the current CATIA drawing area?

Answer: select TOOLS - > OPTIONS - > VISUALIZATION -- - > DISPLAY CURRENT SCALE IN PARALLEL, MODE option. The value of the lower right of the screen represents the DISPLAY ratio of the geometry IN the CURRENT drawing area.

2: how to quickly define the direction of the sketch?

Answer: press CTRL key to select the edge of the X axis in the sketch, then select the sketch plane, then select the sketch function. The sketch will be automatically transferred to the desired direction.

3. How to adjust the size dynamically in SKETCHER?

Answer: select the already marked entity, then hold down the SHIFT key, then drag the entity, and the dimension value marked on the entity will change dynamically.

4: when starting CATIA, there will always be a new CATPRODUCT file, how can you take refuge?

A: at the START - > ALL PROGRAMES - > TOOLS - > ENVIRONMENT EDITOR V5R * * create a new ENVIRONMENT variable, NAME: CATNOSTARTDOCUMENTVALUE: YES you can change this kind of problem.

5: some MODEL files from CATIA, the information of PROJECTCONFLICT when read, how can I read it?

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A: in the directory of \$HOME, modify the file of usrenv.dcls to add ATSITE, PRJMODEL = TRUE. This is the line. (UNIX environment)

6: how do you skip the animation to speed up the display when you enter the CATIA sketcher workbench?

Answer: TOOLS - > OPTIONS - > DISPLAY - > NAVIGATION,

UNCHECK "ANIMATION DURINGVIEWPOINT MODIFICATION". You can disable ANIMATION.

7: how to change the size and color of the default coordinates of the system?

Answer: TOOLS - > OPTIONS - > INFRASTRUCTURE - > DISPLAY IN GEOMETRY AREA - > DISPLAY IN GEOMETRY AREA - > AXISSYSTEM DISPLAY SIZE (IN MM), change the default value from 10 to 40.60, etc., can change the SIZE of the datum level, but the color can only be changed directly on the GRAPHIC PROPERTIES tool on the interface.

8: how to switch between feature trees and entities?

Answer: you know may be in the feature tree click or click on the lower right corner of the screen coordinate system. Here I introduce a method for the premise is that your mouse is three key) hold down CTRL and then slide on the mouse wheel with respect to OK.

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9: what if your three-button mouse is suddenly broken and the family has only two keys? I have a good idea, which is actually the function of CATIA.

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Answer: open tool-> OPTION - > DEVICES AND VIRTUAL REALITY; Then you can edit it in TABEL SUPPORT, and see the list of features on your interface so you can continue to learn.

10: when you open multiple documents in CATIA, do you have a quick conversion window?

Answer: hold down the CTRL key and then press the TAB key

to quickly convert the window.

11: how to quickly scale in CATIA?

Answer: press the CTRL key and then press the mouse button to enlarge the zoom. Press the mouse button before pressing the CTRL key to rotate.

12: how to temporarily hide command dialog box during command execution?

A: click the button on the right side of the screen right corner to hide.

13: there is no quick way to get the same parts in the assembly.

A: under the current assembly file, use the INSERT first - > EXISTING COMPONENT to the needed parts, and then select the parts, the click INSERT - > FAST MULTI INSTANTIATE can quickly call again. Can also be defined in DEFINE MULTI INSTANTIATE a call in the same part number, implement a transfer into more of the same parts.

CATIA's unique skill

1. If you want to select elements that overlap, use ALT + middle key to select
2. The use of the search function, the model (assembly) does a certain degree of complexity, and there must be a

lot of points, lines, faces and so on. How can they be hidden? Search first, then select, and finally operate

3. Complex assembly will appear all sorts of puzzling problems, such as in the current work condition of the parts do not choose other parts, etc., can first consider the isolate (right click to find!)

4. Add the constraint, mark the button directly with the size, and then right-click on the two elements, and then select the appropriate constraint more quickly than that constraint

5. The movement of the parts in the assembly and the movement of the body in the part of the assembly, using the compass, right-click the compass to select edit, set the value of movement and rotation, and place the compass on the moving parts, which is convenient

6. How do multiple machines share a working environment (setting or option) in CAITA and how to share an engineering drawing standard? Place the environment and criteria on the server and set the environment path. Of course this requires administrator rights

7. How complex can caita do with motion simulation? You can say that, in addition to the non-linear motion (which is very rare), CATIA can do it, and you can see the sequence of the sequence

In fact, there is a way to choose a few elements that

overlap

You can put the mouse in the general position of the object you want to select,

Then try to press the four arrow keys on your keyboard, and a browsing window appears near the mouse

Now you can choose which element you want by pressing the keyboard, which is CATIA's only secret.

CATIA shortcuts

Ctrl + Page up -- -- zoom in

Ctrl + Page down -- -- zoom out

Alt + shift + up and down arrow - rotate

Ctrl + up and down arrow -- pan

Ctrl + shift + left and right arrow rotate

Shift + F3 -- work on specification tree

Shift + F2 -- specification tree overview

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CTRL + o -- -- -- -- -- -- -- -- to open

Alt + Enter -- - property

CTRL + F -- -- -- -- -- -- -- -- to find

The most important one:

CTRL + s - - don't forget to keep it

Some useful parameters for CATIA environment variables!

CATLM_ODTS = 1 - no license error is displayed when booting

CATNoStartDocument = no - starts without product

CNEXTBACKGROUND = no - the background color is not
generated when started

CNEXTSPLASHSCREEN = no - the ds star is not displayed at
startup

L_WILSON_LAN = 1 - supports the Wilson curve generation

CGM_ROLLINGOFFSET = 1 - can rotate offset in GSD mode

TAILLE_MEMOIRE_CHOISIE = 1 - IGES optimization output

SHOW_CST_CHILDREN = 1 -- in the sketch drawing, the father-level note emphasizes the display

CNEXTOUTPUT = console -- displays the CATIA logo in the Windows command

MM_NO_REPLACE = 1 - unconnected component assembly reset

IRD_PRODUCTTOPART = 1 assembly mode to hide

PRINT_WITHOUT_DEPTH = 1 - remove print (V5R10)

MFG_MULTI_MP_APPLY = 1 - multi-instance manufacturing process

MFG_CATMFG_REMOVE_MOTION_TOOL_CHANGE = 1 - remove to X Y Z

To start > catia > tools > Envoironment edit catia's environment variable editor, add it

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CATIA fast express key standards (CATK II) CATIA command shortcuts Note is often used

Sketcher will F2

Positioned Sketcher will F4

Rofile F5

Rectangel F6

Circle F7

The Line F8

The Quick Trim F9

The Constraint F10

The Constraint... F11

Flt all in Home Exit Workbench End Pad Hole Edge Fillet
Chamfer Pocket F9

Rectangular Pattern F6

Zoom Area F5

Full S

CATIA 快捷键标准 (CATK II)

舞台布景设计者 F2

定位舞台布景设计者 F4

Rofile F5

Rectangel F6

圆 F7

行 F8

快速修剪 F9

约束 F10

约束...到

外语教学都在家里

退出工作台结束

板孔

边缘角

倒角

口袋 F9

矩形模式 F6

缩放区域 F5

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角 F12

PartDesign Alt + P

舞台布景设计者 Alt +年代

大会 Alt +

Drafring Alt + D

F3

重复上一次操作

Ctrl + Y

实时放大缩小