CATIAv5 Tips and Tricks

Selections & Settings:
How can you make Multi-Selections with Power Impute Method?

- prompt in command bar:
  - **t:**   for feature type
  - **n:**   for feature name
  - **color:**   for entity color

- examples

  For selection of all holes features type:
   t:hole

  For selection of all features named "body":
   n:body

  For selection of all sea black color objects:
  color:'sea green'

  For selection of all features of type pad named housing:
  t:pad & n:housing

How can you keep/delete your toolbars settings:

To keep all toolbars settings:
   Tools/Customize/Toolbars/Restore All
In Application Data/Dassault Systems/CATSettings folder make "Dialog Position CATSettings" read only.

To delete all settings and start with default CATIA settings:
   In…/Application Data/Dassault Systems/CATSettings/[…]   select all and delete all files

Drawing

How can you use Master Drawing with Title Block in the Background View:

Edit/Background View//Edit + Working Views to toggle

File/Page Setup then "Insert Background View" then prompt to open drawing containing the background view (title block) that you want to copy.

Tools/Options/Mechanical Design/Drafting/Layout and select "Copy background view" and "first sheet" means when you insert a new sheet, a Title Block is automatically inserted.

How can you insert text file in Drawing:

Windows text only (Excel, Word):
Insert/object/From File then select the type of file (Excel, Word, etc.,)

How can you visualize virtual Intersection in Drawing:

To visualize a virtual intersection in a drawing (for example tangency fillet intersection) one of the best solutions is to make them reference geometry in sketch.

How can you create a different size for paper in Drawing:

Type in drawing size: 2A0 then modify length or width.

Sketcher
How can you insert sketcher tangency direction for spline:

Select the spline Control Point, then right-click and select in contextual menu "Tangency" for controlling tangency visualize by a red arrow. Then constrain the red arrow parallel or collinear to the entity you choose to control the tangency.

How can you define h and v axis for sketch?

Multi-select (Ctrl task) two edges on an opened part, the axis is aligned on these two edges. Select a surface and then select the Sketcher icon.

Select one plane of the local axis, h and v are aligned to the main axes of this selected plane. Association is kept between both the plane and the sketch.

If you select one or two edges, h will be parallel to the projection of the first edge in the sketch plane.

If the two projected edges appear as two lines that intersect in the sketch plane, the origin of the sketch axis will correspond to the intersection of the two lines. Note that this origin point is a construction point used as a positioning assistant and it cannot be modified.

How can you use the Sub-Elements of a Sketch:

For Pad and Pocket Features:
In Pad (Pocket) Definition dialog box click the Selection field then right-click and select "Go to Profile Definition" contextual command and the Profile Definition dialog box is displayed. You can define whether you need the Whole geometry, or sub-elements only. Select the sub-elements for profile and click Remove to remove the associated profile from the selection, then click OK to validate selection. If you encounter complex profile causing ambiguity cases, the application lets you determine which lines you want to use.

For Other Sketch-Based Features:
You can use some elements of a sketch with Extract Multiple Edges command from GSD toolbar. This operation creates a new feature named "Extract" and the specification tree indicates this creation. You may select this feature (like sub-elements of a sketch) for creation other Sketch-Based features.

Part Modeling

How can you create a loft feature with Start and/or End section as point section?

For loft features you cannot use as section a point but you can use Relimitation options for create this kind of geometry.

The Relimitation tab lets you specify the loft relimitation type. You can choose to limit the loft only on the Start section, only on the End section, on both, or on none (default option):

- when one or both are checked: the loft is limited to corresponding section
- when one or both are unchecked: the loft is swept along the spine:
    - if the spine is a user spine, the loft is limited by the spine extremities
    - if the spine is an automatically computed spine, and no guide is selected: the loft is limited by the start and end sections
    - if the spine is an automatically computed spine, and guides are selected: the loft is limited by the guides extremities.

With this last type of loft relimitation, you can compute a loft geometry with

guides which intersects in a point. In that case your geometry can convert in a point (for start or/and end geometry).

Assembly Design

How can you use multiple configurations of the same part in an assembly?
If you want different configurations (Design Table) of the same part in the assembly you have three options:

1. If they are the same, instance the parts.
2. If they are unique, and but very similar, use Copy + Paste Special + Break Link to get a new part reference exactly like the original. Modify this new part reference. It's a different part with a different part number.
3. If it's unique but you'd like to add new features to the original, create a new part and Copy + Paste Special + AsResultWithLink the part body from the original to the new. This will create a parent-child relationship between the original and the new. Add additional features to the new part. When you modify the original, the new child part will also update.