Work Instruction

015 Research + Development

Basics and Methodology CATIA V5 - Sketcher

W_015_013

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1 Purpose

The purpose of this work instruction is to optimise the management and quality of the data generated by CATIA V5 at GRAMMER AG. A uniform procedure and structure for both data generation and data management is intended to increase efficiency in the application of CATIA V5. Moreover, data should be created using the new, uniform procedure and also structured and managed in such a way that its usability within the process chain is ensured. The subsequent processes shall draw the maximum benefit from the data provided.

2 Scope

This work instruction shall be binding for GRAMMER AG as well as for external partners that are part of GRAMMER AG's design/development process and create data for GRAMMER AG in this context or provide GRAMMER AG with such data.

This work instruction contains specifications for working with the CAD system CATIA version 5 (CATIA V5) in GRAMMER AG's Design department as well as in cooperation with external design and engineering partners that generate data for GRAMMER AG.

This work instruction as well as the methods and procedures described in it shall also apply to customer projects unless replaced completely or in part by separate agreements.

New knowledge regarding the handling of design tasks and design orders with CATIA V5 has been incorporated into this work instruction. Alongside this work instruction, the attachment “A_015_012_Sammlung_CAD_PDM_Dokumente_Collection_CAD_PDM_Documents” shall also be binding.

The following applies to the application of the new work instruction and the expiry of the previous work instructions:

- For “ongoing old projects”, the rules specifically defined for that project shall apply. If such rules are missing completely or in part, this work instruction shall apply where necessary.
- If there are definition gaps, in case of doubt the rules applicable up to now as well as implemented methods shall apply in these situations.

3 Terminology

CATIA/SmarTeam-specific descriptions of functions are written in italics.

CAx Data	Umbrella term for data/files that are created in a **c**omputer-**a**ided system, e.g. CATIA.
CATPart	CATIA V5 part (single part)
CATProduct	CATIA V5 product (assembly of CATParts and subassemblies)
CATDrawing	CATIA V5 drawing (comprising one or more drawing sheets)
Component	CATIA V5 structuring element in the specification tree (without own physical document, e.g. on hard disk), as a representation either of a part without a geometric counterpart or of a product comprising CATParts, CATProducts or Components
Root Product	CATProduct that includes several subassemblies (CATProducts)
SmarTeam	Software for managing CAx data documents from CATIA V5
PDM	Product Data Management
PLM	Product Lifecycle Management
Viewer	Software for viewing 2D images and 3D representations

4 Responsibilities

The implementation of and compliance with this work instruction are the responsibility of the vice president R&D.

5 Description

5.1 General information

Definition of the Sketcher:
The Sketcher is a plane-oriented (2D) tool. The Sketcher generates wire-frame geometry and has the ability to generate and manage parametric profiles in an associative design environment.

The Sketcher is the central tool and foundation for the following CATIA V5 applications and modules:

- Volume design in the PartDesign module
- 2D -> 3D migration process
- Interactive drafting
- Sheet metal design
- A few surface generation modules

It is usually the first tool that a new user learns and uses or that a designer employs at the start of a project.

Additional features of the Sketcher:
The Sketcher is a tool that is easy to learn and operate.

5.1.1 Scope and information on this document

This document describes and explains the basic role that the Sketcher plays in drafting and amending. This document should provide users with a work of reference for everyday use.

The following points are addressed in this document:

- Introduction to the Sketcher
  - Key concepts
  - Basic principles of sketch creation

- Feature design with the Sketcher
  - What should be contained in profiles
  - Use/integration of 3D (reference) geometries
  - Other geometry creation tools

- Constraints and parameters
  - Automatic constraint allocation
  - Relationships
  - Stable relationships
  - Naming and organisation

- Link management of sketches
  - Re-use of sketches
  - External connections & links
  - Use of edges

- Sketch analysis - identifying and resolving problems within sketches
  - Profile routing
  - Non-closed profiles / gaps
  - Design / standard geometry

5.2 List of general questions about working with the Sketcher

A few questions are posed below as a small test to find out the level of knowledge when it comes to using the Sketcher. An experienced user should be able to answer these questions:

1. What is meant by the terms under-constrained, over-constrained and fully constrained?
2. How do you know whether a sketch is fully constrained?
3. How many rotational axes can you get actively in a sketch?
4. Can you use “open” (non-closed) profiles?
5. Can profiles be used for several/different features?
6. When are fillets and chamfers incorporated into sketches?
7. What is a sketch support?
8. How can you re-position the axis system H, V?
9. How do you drag/generate a hexagon?
10. How can you see the other/reverse side of the sketch?
11. Can you use a sketch in a different part?

5.3 Basic principles of the Sketcher

In terms of data processing, files in the CATPart format form the basis for sketches. It is in these that sketches are created and managed. For GRAMMER AG specifically, all CAD documents for CATIA V5 that are to be managed in SmarTeam must be generated using the script-based “*New Documents*” function in CATIA V5. This is the only way to guarantee that:

- A test number or SAP number is generated automatically.
- Filenames receive the SAP-compliant designation / part number.
- The right starting model has been chosen.
- The correct drawing standard has been activated.
- A unique SmarTeam ID is allocated.

Data that is not generated according to this method and does not conform to GRAMMER AG’s specifications must not be used or checked into SmarTeam.

5.3.1 Calling up the Sketcher

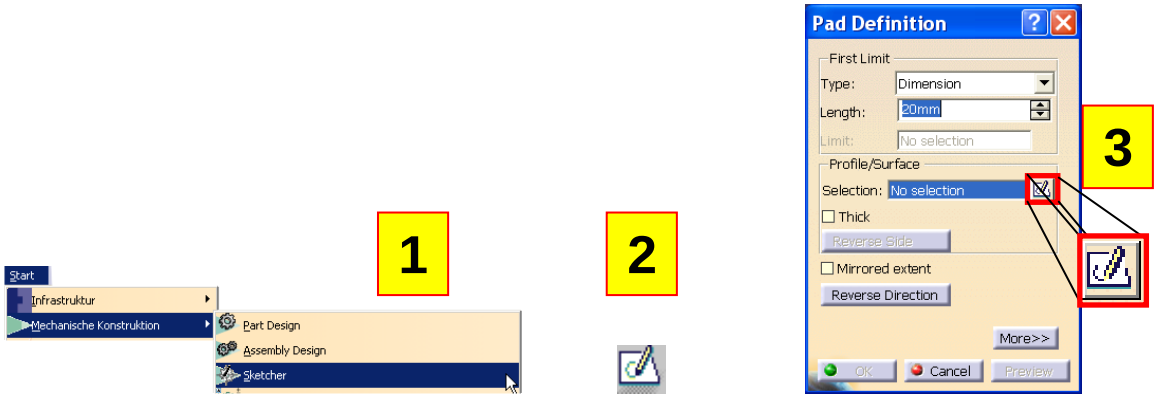
If a new single part file has been generated as described above, the Sketcher can be launched in one of the following ways:

1. From the toolbar: *Start – Mechanical Design – Sketcher*: This launches the Sketcher module -> then select a design plane on which the sketch is to be created.

2. Using the function button: In most design environments for the creation of single part geometry this is located in the workbench.

3. Interactively using a "*sketch-based feature*" function.

The methods listed here are illustrated below:



After applying the respective method, the user must select a plane or a planar face/surface on which the sketch shall be dropped and defined.

If a file already contains a sketch, this sketch can be edited by following method 2. or 3. shown on this page. In addition, it is always possible to edit existing sketches (or other feature elements as well) by *double-clicking*.

5.3.1.1 Types of sketch

For generation purposes, CATIA V5 distinguishes between two types of sketch:

Sliding: If a sliding (non-positioned) sketch is generated, the reference is always to the current, absolute axis system. The orientation of the sketch is defined by the current axis system.

Positioned: This generates a positioned sketch in which the origin and orientation are specified explicitly.

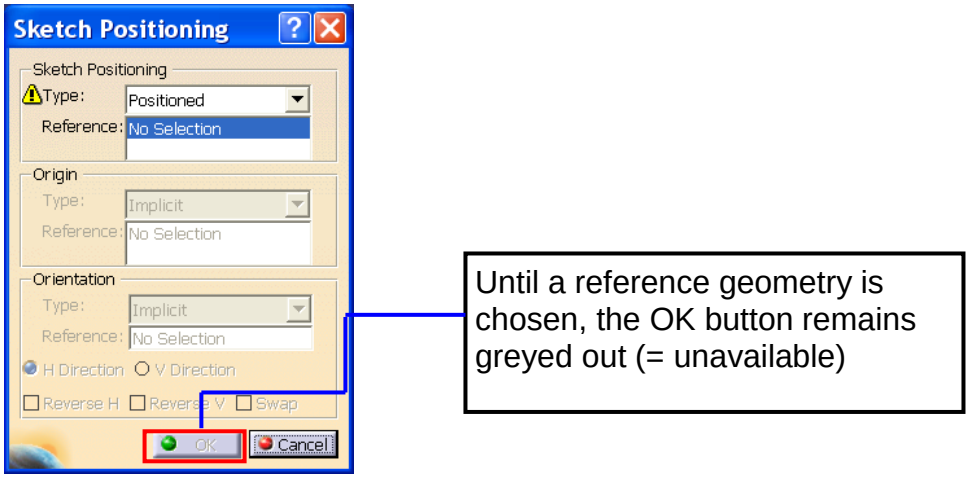
5.3.2 Sketch support

Sketch supports are understood as the support elements onto which the geometry of the sketches is dropped. Sketches are 2-dimensional and can therefore only be defined on planes or planar surfaces or faces.

In principle, a sketch always needs a support for its definition or construction. The only exceptions to this rule are isolated sketches. These are either existing sketches that have subsequently been isolated from their support or sketches resulting from data migration. The generation of isolated sketches is described under *5.3.3 Positioned sketches*.

5.3.3 Positioned sketches

To generate a positioned sketch, the module is launched using the function button . This opens the following window:



Details concerning the position and orientation of the sketch can now be specified individually in the dialog window.

If the setting for the sketch type is configured to *Sliding* in the dialog window, the symbol for the sketch object is automatically adjusted to .

<u>Important:</u> It is not mandatory to complete all the fields in this window in order to generate the sketch. Only the following details have to be supplied:

- *Type of sketch:* This should remain as ***Positioned.***
- *Reference:* Plane or planar surface onto which the sketch shall be dropped.

Optional details

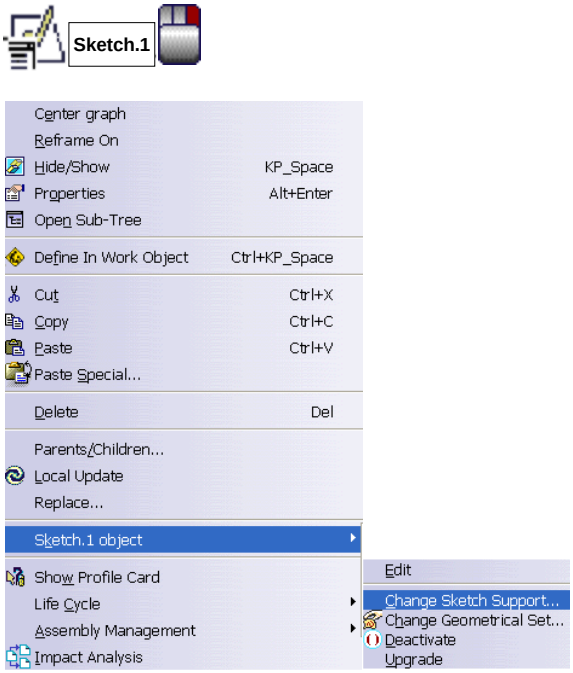
- *Origin:* on which the sketch-internal axis system is to be positioned. If this is not specified, the orientation is always based on the absolute origin of the part.
- *Orientation*: axis, line, surface, etc., to which one of the sketch axes should be oriented.

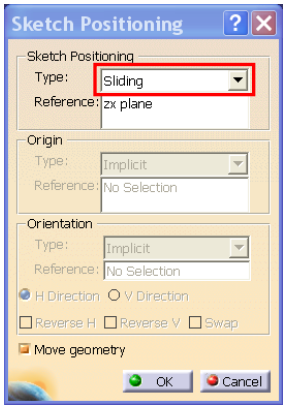
Once all the relevant entries have been made, the dialog must be confirmed by clicking on OK. This takes the user to the sketch mode with access to the workbench for creating, manipulating, analysing and parameterising 2D profiles.

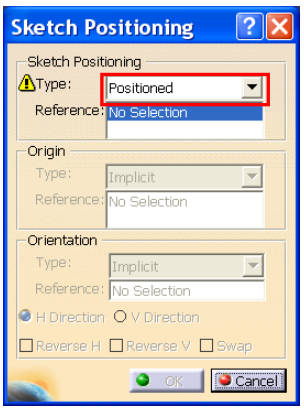
5.3.3.1 Converting sketches into positioned sketches

In principle, any “common, non-positioned sketch” can be converted into a positioned sketch. The function for this can only be accessed via the RMB shortcut menu. Take the following steps:

- Expand the structure tree, if necessary
- Position the mouse button on the corresponding sketch symbol of the structure and open the shortcut menu with the RMB.
- Select menu item *Sketch.xx object* -> Change Sketch Support …
- In the dialog window, configure the setting as shown below.



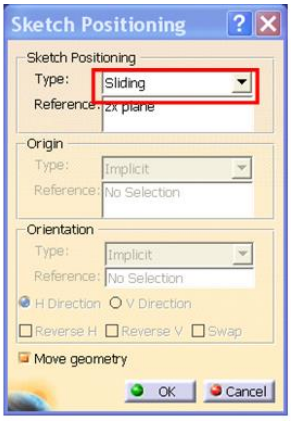


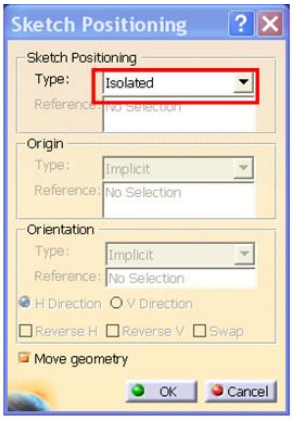


5.3.3.2 Isolating sketches

In principle, any sketch can be isolated from its reference (plane, planar surface). Isolation is carried out as follows using the RMB shortcut menu:

- Expand the structure tree, if necessary
- Position the mouse button on the corresponding sketch symbol of the structure and open the shortcut menu with the RMB.
- Select menu item *Sketch.xx object* -> Change Sketch Support …
- In the dialog window, select the setting ***Isolated*** under the Type option and confirm the dialog by clicking on OK.





This procedure gives the structure element of the sketch the appearance of a positioned sketch. The symbol (which precedes the name of the sketch) is shown as follows:

Note: An isolated sketch can be converted back into a reference-dependent sketch at any time using the method described in the section “Converting sketches into positioned sketches”.

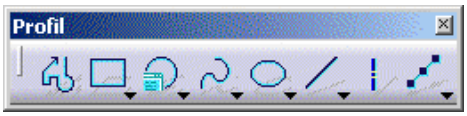
5.4 Work environment of the Sketcher

The work environment of the Sketcher is divided, in terms of its workbench, into the following toolbars:

- Geometry-generating toolbars (***Profile***)
- Geometry-manipulating and/or duplicating toolbars (***Operations***)
- Toolbars for parametrics and geometry control (***Constraint***)

These toolbars, together with the "*Sketch tools*" and "*Visualization*" toolbars, are the most important functions and application areas in the Sketcher.

The table below presents these toolbars and provides a brief explanation of each one.

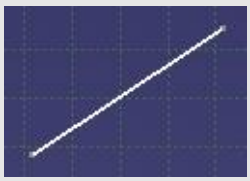
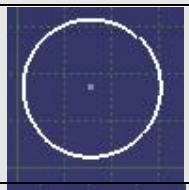
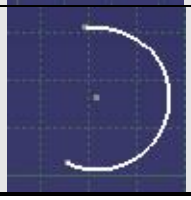
Toolbar	Description	Information
	Functions for generating contours and profiles, predefined profiles, such as rectangle, hexagon, circle, ellipse or spline, and points.	Functions from this toolbar only create geometry. So, when constructing a sketch, this is the function area that is used first.
	Tools to generate radii, chamfers or edges at the intersection of two lines. Other editing functions for trimming, referencing 3D geometry or mirroring elements, for example.	The functions in this toolbar can be described with the umbrella term "manipulation".
	Functions for generating dimensions manually or automatically or for animating or grouping existing constraints.	Functions in this toolbar are there to control and manipulate the geometry elements generated.
	Tools to activate (orange) or deactivate the grid, to generate design geometry or to identify constraints automatically.	Manages the work environment and behaviour of the Sketcher. Since this toolbar expands to the right, depending on the active function, it should not be docked and should be positioned as far as possible towards the left-hand edge of the window.
	Functions for configuring visualisations. Provides different presentation options.	Manages the presentation and, to some extent, the behaviour of the Sketcher.

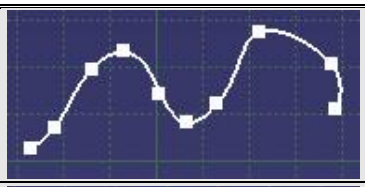
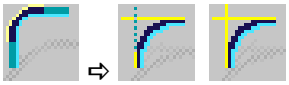
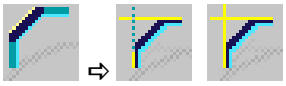
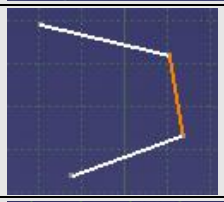
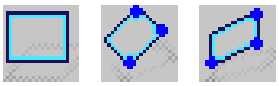
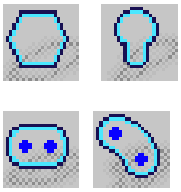
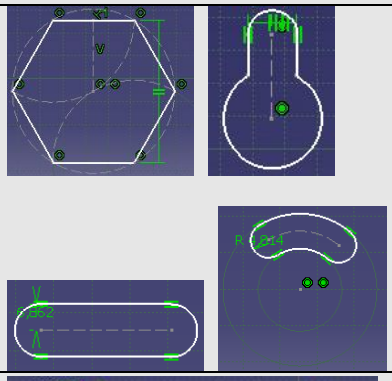
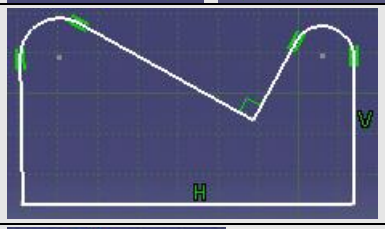
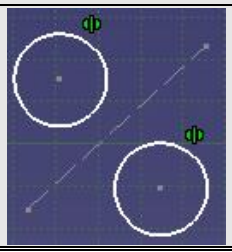
5.5 Generating geometries

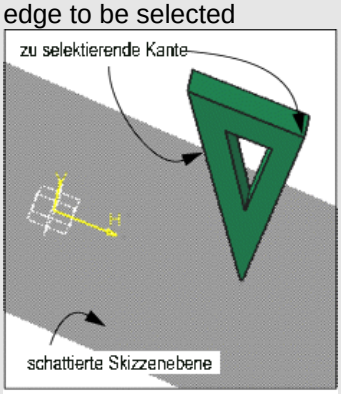
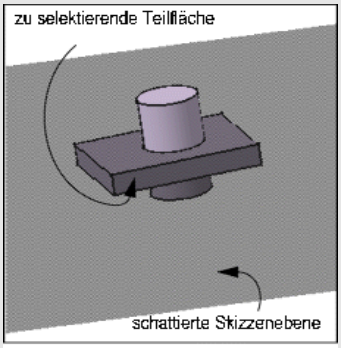
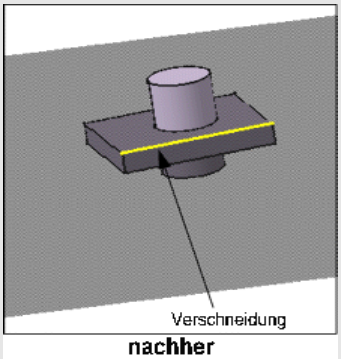
The Sketcher is used to create 2-dimensional profiles. A distinction is made here between open and closed profiles. Use of these types of sketch depends on the application (volume or surface design). It is not possible to generalise and say that open profiles are for surface design and closed ones are for volume design because application is mixed.

5.5.1 Types of geometry

The following table provides an overview of the possibilities for 2D geometry generation. This is a list of the most common functions; it is not intended to be exhaustive.

Element / button	Geometrical representation	Information / explanations
Points:		If the sketch is used for a volume definition, points must be present as design geometry.
Line:		Lines can be defined as a segment or with an infinite length. Line definition is controlled during creation via the toolbar.
Axis (rotational axis):		There can only ever be one rotational axis per sketch. **<u>Attention:</u>** This function is often confused - it is only intended for the construction of sketches that are used later as rotational geometry (e.g. shaft & groove function).
Circles:		Constant curvature
Arc & circle segment:		Constant curvature

Splines:		Constant curvature or irregular curvature
Corners (fillet): 		Constant curvature When generating corners, use the Sketch tools toolbar to influence trimming.
Chamfers: 		When generating chamfers, use the Sketch tools toolbar to influence trimming and the dimensioning mode.
Generate rectangles or parallelograms: 		
Generate profiles: 		These geometries are generated using auxiliary geometry elements.
Complete sketching of profiles:		Geometry generation (switching between line, tangent arc and three-point arc) should be controlled at "*beginner*" level using the *Sketch tools* toolbar.
Generate symmetrical elements:		Do <u>not</u> use the *Axes* function for the symmetry line. Generate a line that is defined as a design element.

Project 3D elements onto the sketch plane:	edge to be selected  **vorher** shaded sketch plane **before**	The generated geometry is linked to the reference element. A subsequent isolation of the reference geometry is possible at any time.
Intersect 3D elements with the sketch plane:	face to be selected  **vorher** shaded sketch plane **before**  **nachher** intersection **after**	The generated geometry is linked to the reference element. A subsequent isolation of the reference geometry is possible at any time.

5.6 Defining constraints

Constraints are *specifications* for elements. They are divided into two groups:

- Geometric constraint
- Dimensional constraint

5.6.1 Types of constraint

The following table provides an overview of the possibilities for constraint generation.

5.6.1.1 Geometric constraints

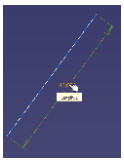
A **geometric constraint** involves a relationship between one or several geometric elements that restricts geometric degrees of freedom by specifying certain element properties. For example, a constraint can specify that two lines run parallel. If three lines are selected or two lines and a point, these elements are automatically arranged parallel to one another, as shown in the table below.

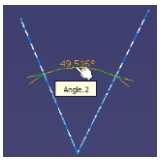
A constraint can be specified for one element or between two or more elements.

Number of elements	Corresponding geometric constraints
One element	Fix Horizontal Vertical
Two elements	Coincidence Concentricity Tangency Parallelism Midpoint Perpendicularity
Three elements	Symmetry Equidistant point

When generating a constraint, note that a green constraint represents a valid constraint as standard. Conversely, a yellow constraint indicates that the corresponding definition is not valid. The software enables the adjustment of colours and of the general representation of the constraints used.

When the cursor is placed on constraint symbols, the software highlights the elements belonging to the constraint system. Here are two possible examples:





5.6.1.2 Dimensional constraints

A dimensional constraint is a constraint whose value specifies the dimensions of the geometric object (such as the length of a line or the distance between two points).

The constraint command completes the profile. The user can specify dimensional constraints or geometric constraints using the constraint command. However, it is usually dimensional constraints that are specified.

Dimensional constraints can also be combined to specify constraints for a component or sketch.

A dimensional constraint can be specified for one element or between two elements.

Number of elements	**Corresponding dimensional constraints**
One element	Length Radius/diameter Semimajor axis Semiminor axis
Two elements	Distance Angle
A diameter constraint can be applied between two lines if one of these lines is an *axis*.	

5.6.2 Constraint representation & symbols

Constraint symbols - representation in the structure tree

Symbol	Description
	Constraints for length, radius, semimajor axis and semiminor axis
	Fix
	Angle
	Symmetry
	Equidistant point
	Perpendicularity
	Coincidence
	Parallelism
	Distance
	Midpoint, concentricity
	Tangency

The table below lists the symbols used to denote the different types of constraint. These symbols refer to the representation on the graphical user interface.

Symbol	Type of constraint
	Rectangular
/	Coincidence / concentricity
V	Vertical
H	Horizontal
	Fix elements / objects
	Tangency
	Midpoint
	Parallel
R 25 / D 50	Radius / diameter
112,5	Distance / length
135°	Angle

5.7 Sketch analysis

If a sketch has been created/edited and problems arise with this sketch (e.g. when defining 3D feature geometry or when leaving the sketch environment), the cause of any errors often lies in a mistake with the geometric or parametric definition of the contour. Given that it can take a lot of time and effort to search for errors, depending on the complexity of the defined geometry, the sketch environment contains tools for analysing sketches.

The function buttons for analysing sketch objects can be found also in the *Tools toolbar* (within the Sketcher environment).

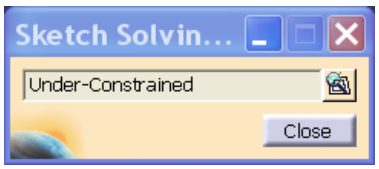
These are described in detail in the points *Quick diagnosis of a sketch geometry* and *Detailed analysis of a sketch geometry*.

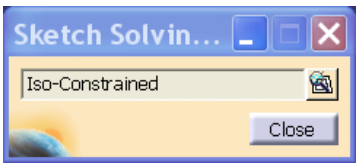
5.7.1 Quick diagnosis of a sketch geometry

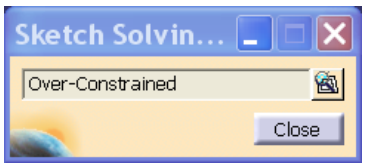
The "*Sketch Solving Status*" function can be used to examine the sketch in terms of the parametrics (executed). This rough analysis solely provides feedback on the degree of parameterisation.

The function can only be called up using the function button (located in the Tools toolbar).

Depending on how the sketch is defined, three statuses can be identified and reported by this tool:







After that, an advanced analysis can be run directly if the sketch analysis function button is selected in the results window. This is used to call up the function described under point 5.7.2.

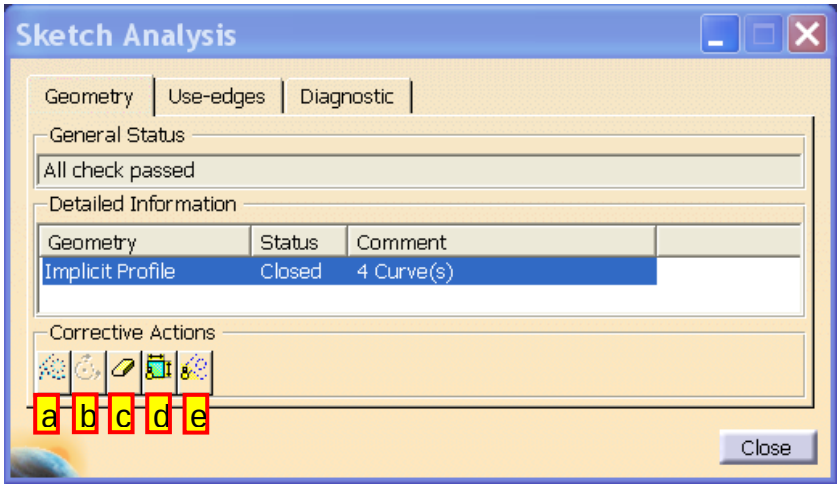
5.7.2 Detailed analysis of a sketch geometry

The "*Sketch Analysis*" window allows a detailed analysis of a sketch and the elements it contains. In addition, functions can be executed to remedy any faults or errors in a sketch geometry. Possible errors that can be identified by the sketch analysis:

- Are contours/profiles closed?
- Overlapping of elements
- Errors in projection
- Duplicated geometry elements (one lying on top of another)

The window for the detailed sketch analysis is opened either by selecting the *Sketch Analysis* button (located in the "*Tools*" toolbar or in the results window of the *Sketch Solving Status* function) or, as a further option for calling up the function, by choosing the menu function under *Tools -> Sketch Analysis*. The window has three tabs in which partial results of the analysis are displayed: "*Geometry*", "*Projections/Intersections*" and "*Diagnostic*".

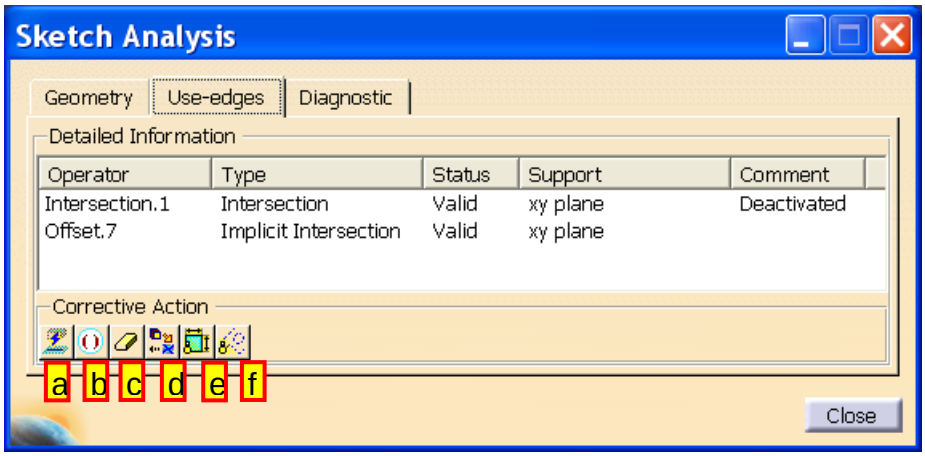
Description of the Geometry dialog window:



General Status:	Status of the function analysis
Detailed Information:	Detailed status including comments on each geometry object in the sketch
Corrective Actions:	If an analysed object has been selected in the detailed information, corresponding actions (e.g. for correction) can be implemented. The following is possible: a. *Set in Construction Mode* -> change the (standard) element to a design element b. *Close Opened Profile* -> close a non-closed profile c. *Delete Geometry* -> delete a (disruptive) element d. *Hide Constraints* -> hide/show all sketch constraints e. *Hide Constructive Geometry* -> all geometry objects with the attribute *design element* in the sketch are hidden/shown

Description of the Projections/Intersections dialog window:

Any existing projections and intersections (including constraint references) with 3D geometry are displayed in this dialog window.



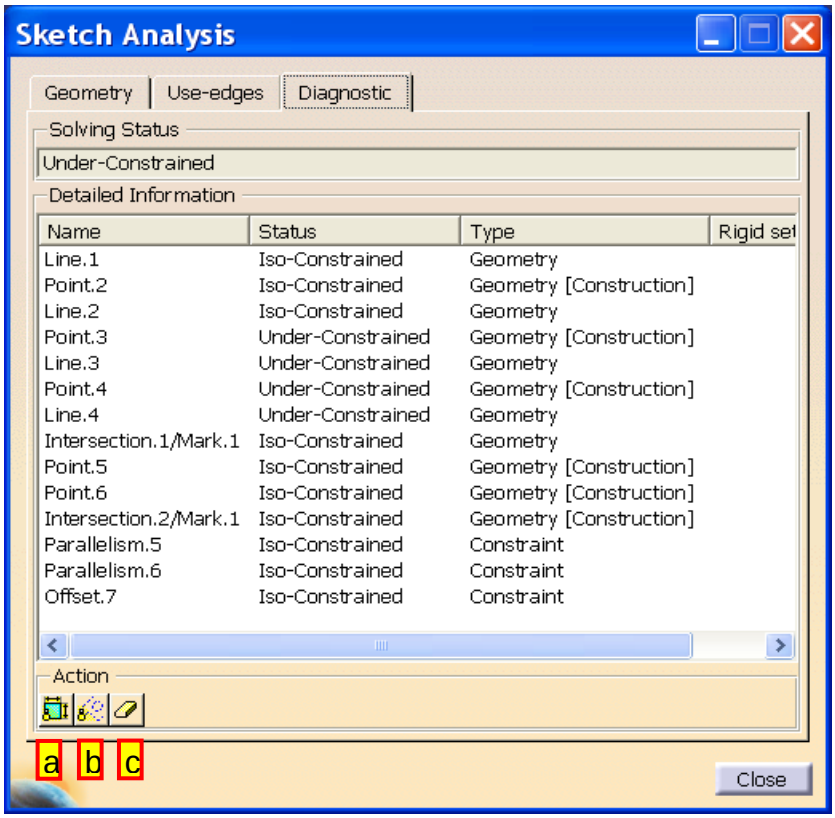
General Status:	Sketch constraints
Detailed Information:	Detailed status including comments on each geometry object affected in the sketch
Corrective Actions:	If an analysed object has been selected in the detailed information, corresponding actions (e.g. for correction) can be implemented. The following actions are possible: a. *Isolate Geometry* -> isolate the reference element b. *Activate/Deactivate* -> activate/deactivate the reference c. *Delete Geometry* -> delete a reference/link or a resulting object d. *Replace 3D Geometry* -> create a reference to a different 3D reference object e. *Hide Constraints -> hide/show all sketch constraints* f. *Hide Constructive Geometry* -> all geometry objects with the attribute *design element* in the sketch are hidden/shown

Description of the Diagnostic dialog window:

The *Diagnostic* tab provides a basic summary for the sketch and a list of all the objects within the sketch. It explicitly indicates, among other things, the degrees of freedom of the active sketch and of the geometry and constraint elements contained in it.

In the *Solving Status* field, a general appraisal of the sketch as a whole is made. This appraisal is determined in line with the analysis function *Sketch Solving Status*.

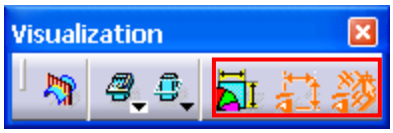
The Detailed Information list provides the user with the status for each geometry and constraint element in the sketch. The degree of constraint is assessed for every object. If a list entry relates to a geometry element, its type (standard element, design element) is also listed.



Solving Status:	Status of the sketch (as a whole)
Detailed Information:	Detailed status including comments on each geometry and constraint object in the sketch
Corrective Actions:	If an analysed object has been selected in the detailed information, corresponding actions can be implemented. The following actions are possible: a. *Hide Constraints -> hide/show all sketch constraints* b. *Hide Constructive Geometry* -> all geometry objects with the attribute *design element* in the sketch are hidden/shown c. *Delete Geometry* -> delete a reference/link or a resulting object

5.7.3 Diagnostic colours

The system has the ability to convey a status to the user directly using the colour of the objects during the generation and parameterisation of elements. Through permanent checking of the object status, combined with colouring of the objects in the diagnostic colours, the user receives interactive support when editing sketches. To activate this function, the framed buttons in the *Visualization* toolbar should be in highlight mode.



This allows the colour of a geometry or constraint element to be obtained directly from the geometry window of the Sketcher.

5.7.3.1 Definition of object colours

An element that has just been created by a user or is currently selected is displayed in the colour **orange**. The colour has no direct significance for the analysis.

A geometry element whose spatial position has not been defined clearly, i.e. it has degrees of freedom, is shown in **white** or **grey**. A standard element is displayed in white, a design element in grey.

If a geometry element has been generated through, for example, a projection or intersection of a 3D geometry with the sketch plane, it cannot be manipulated by the user directly because of its associativity with its 3D reference. In such a situation, the system signals this with the colour **yellow**.

If the position of a geometry element has been defined clearly (the element has no degrees of freedom), the element is shown in **green**.

The colours **brown**, **red** and **pink** are warning colours and alert the user to problems in the sketch.

- All elements that are coloured **pink** indicate that the geometry is over-constrained. Over-constraining can be remedied by purposefully deleting individual constraint elements.

- If CATIA V5 is no longer able to represent a geometric situation because constraint and geometry elements describe an inconsistent geometry or the constraint and geometry elements are not recalculated (in part), these objects are displayed in **red** and **brown**.

Colour code table

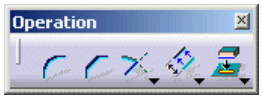
Colour	Meaning
White	Standard element with degrees of freedom
Grey	Design element with degrees of freedom
Orange	Just created/selected element
Yellow	Projected or intersected element (also *protected* element, e.g. axis system)
Green	Fixed element or element with clear position (with no degree of freedom)
Pink	Over-constrained element (too many constraints)
Red	Element inconsistent
Brown	Element not calculated temporarily

5.8 Tips, notes & rules

The general procedure within the Sketcher should be as follows:

1. Define the profile / contours, which should be roughly true to scale but do not yet necessarily have to correspond to the later exact (finished) contour. To do this, use the functions in the "*Profile*" toolbar.

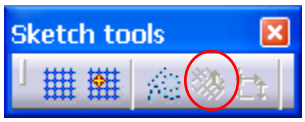
2. Insert other geometry elements and modify (trim) the previously created "basic contour". To do this, use the functions in the "*Operation*" toolbar.



3. Assign dimensional and geometric constraints in order to obtain the "finished" contour. To do this, use the functions in the "*Constraint*" toolbar.

4. Keep the generated sketches simple. Depending on the complexity and structure of the part, fillets, chamfers and holes are inserted using the 3D feature functions. This has the advantage of being more change-friendly and enabling a process description with the help of the feature technology.

Note:
When defining contours and profiles, always pay attention to any element links (geometric constraints), that are automatically proposed and defined by the system via the so-called *Smartpic* function. Such constraints are frequently overlooked when generating contours/profiles. At a later point in time (during dimensioning), these constraints may cause the elements within the sketch to be over-constrained and may make it impossible to make changes using defined measurement parameters. Constraint suggestion can be suppressed temporarily while defining contours by pressing the *Shift* key on the keyboard; alternatively, the button for automatic generation of geometric constraints must be deactivated in the Tools toolbar or under Tools -> Options -> Mechanical Design -> Sketcher.



Sketch tools toolbar – Button for automatic generation of geometric constraints

6 Applicable documents

Attachment “Sammlung CAD-/PDM-Dokumente / Collection CAD / PDM Documents”	A_015_012

7 Changes