

MODELLING SLIDES AND JOINTS IN MACHINE TOOL DOMAIN USING 3D CAD SOFTWARE

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Abstract: Assembly design in mechanical domain imposes to use joint conditions between mechanical parts restricted by functional aspects. The paper presents main features to impose slides and joint conditions using software package CAD/CAM/CAE CATIA.

Key words: CAD, joint condition, CATIA solutions, assembly design.

1. INTRODUCTION

Table 1

Machine tool domain uses frequently mechanical assembly with high complexity and contact restraints. CATIA V5 is a powerful tool to design and manipulate V5 objects and to work with V4 models. The main components in mechanical design domain are: Part Design (PDG), Assembly Design (ASM), Generative Drafting (GDR), Generative Shape Design (GSD), DMU Kinematics Simulator (KIN) and Sheet Metal Design (SHM).

The application in this paper is the mechanical part design and assembly for the milling and boring machine AF 105. NC axis used are defined Z for axial spindle nose translation, X for transversal feed of the table, Y for vertical tailstock slide translation and W axis – cross table displacement. All this four axis are using ball screw spindle and electric drive step by step motors. The mockup obtained is used to dimensioning, verifying and optimising for mechanical parts.

2. SETTING CONSTRAINTS

Assembly Design (ASM) module allows the design of assemblies with an intuitive and flexible user interface.
















Assembly Design considers the following operations:

- Part Design feature insertion;
- Assembling Components together;
- Moving components;
- Constraining components;
- Analyzing and Modifying an Assembly

Setting constraints is useful when assembly product have to be designed. It is necessary to following the recommendations:

- Constraints could be apply only between the child components of the active component;
- Constraints cannot be defined between two geometric elements belonging to the same component;
- Cannot apply a constraint between two components belonging to the same subassembly if this subassembly is not the active component.

Type of constraints

	Planar Face	Sphere	Cylinder	Cone	Circle
Planar Face					
Sphere					
Cylinder					
Cone					
Circle					

Contact-type constraints can be created between two planes or faces.

The common area between the two planes can be a plane (plane contact), a line (line contact) or a point (point contact).

Table 1 shows the elements that can be selected. [10].

An example of contact constraints between ball-screw and nut, used on a laser cutting installation, is taking into consideration.

The procedure to impose the constraints into CATIA V5 software has three steps followed after the parts models definition:

1. Select the icon **Contact Constraint** .

This command is also available from the **Insert** menu.

2. Select the face to be constrained as shown (Fig. 1).

3. Select the second face to be constrained (Fig. 2).

As the contact constraint is created, first component is moved so as to adopt its new position. Green graphic symbols are displayed in the geometry area to indicate that this constraint has been defined (Fig. 3)

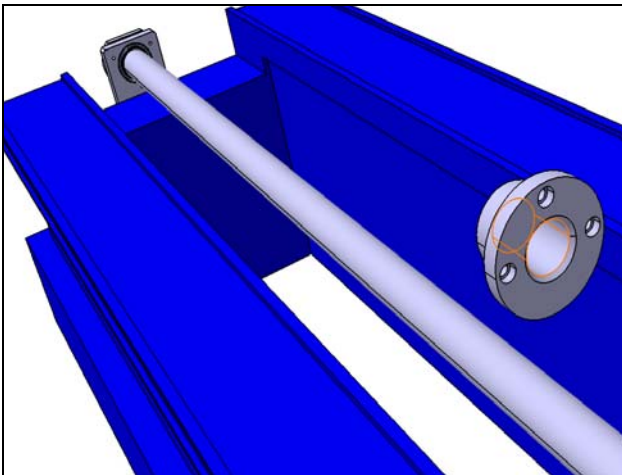


Fig. 1. Two mechanical part before to be constrained.

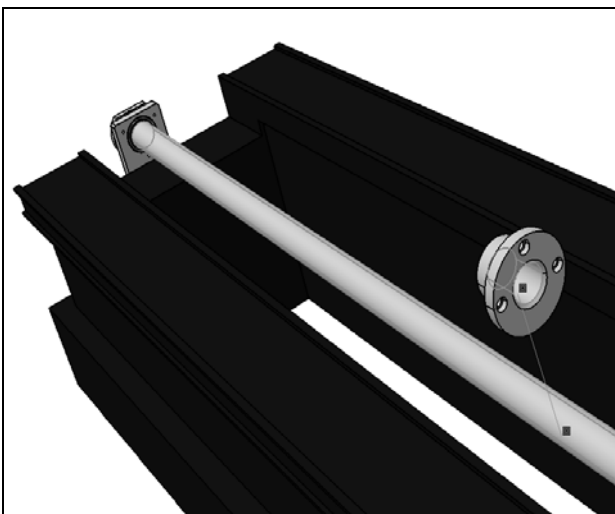


Fig. 2. Two parts to be in contact.

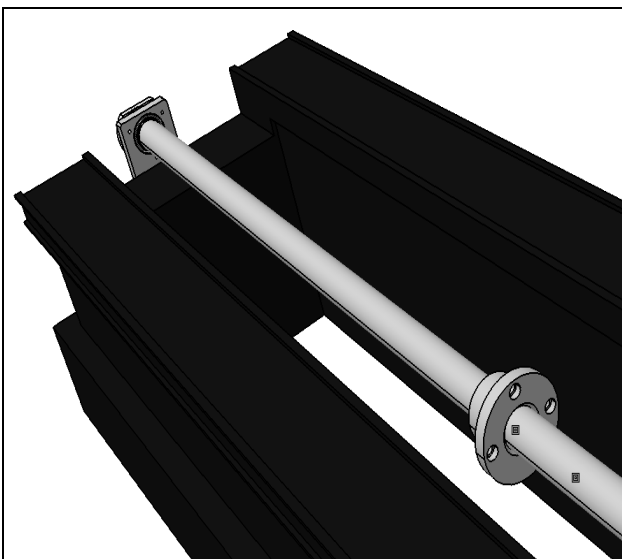


Fig. 3. Ball screw mechanism after constrained action.

This constraint is added to the specification tree too (Fig. 4).

In the Figs. 5 and 6 are presented the other cases when necessary to impose the slide connections mainly is used in the machine tool domain.

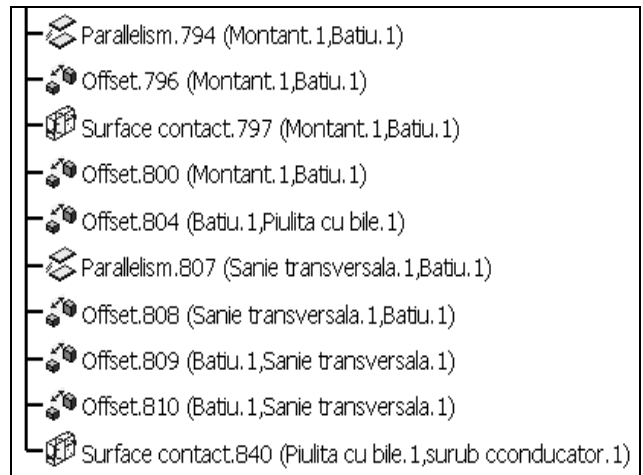


Fig. 4. Specification tree in CATIA menu.

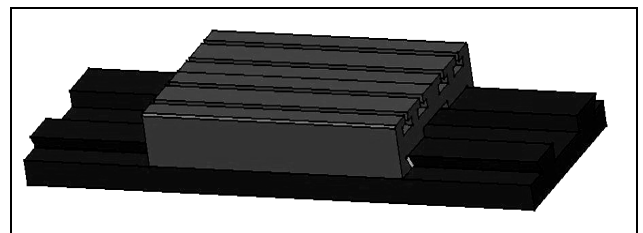


Fig. 5. Example of table contact with plane surfaces.

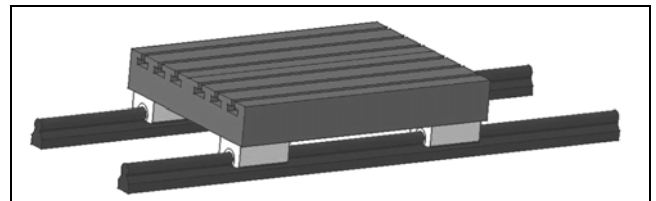


Fig. 6. Example of table contact with cylindrical surfaces.

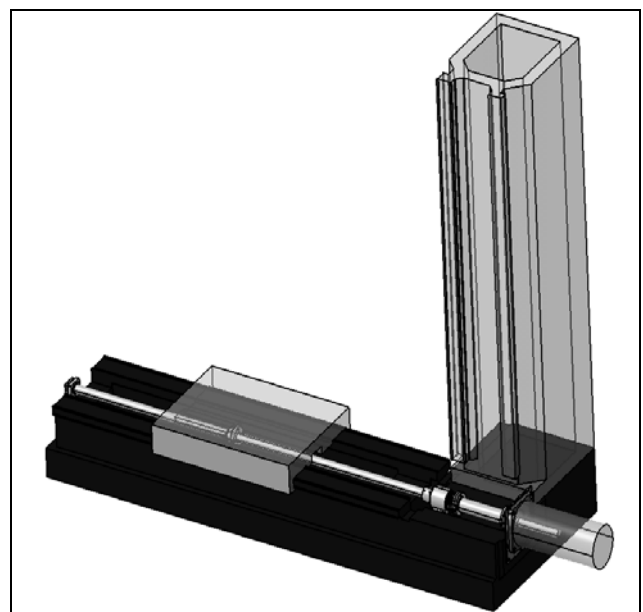


Fig. 7. Assembly equipment designed in CATIA.

For the Assembly Design is created a new feature based on two macros that helps to saving the positioning of ensemble components without saving previously in

Catia V5. These two macros programs were developed being necessary in the practical cases, when CAD applications use in a collaborative way the software Catia V5 and Enovia VPM.

ENOVIA it is the solution for informational management about projection and fabrication piece (n.r-PDM II), included the industrial space modeling, the configuration of digital models, the caption and disseminated information and the process knowledge (to the begin of conception to the sell and exploitation).

The reason of these programs apparition is the following: the Airbus projects usually use for CAD applications Enovia VPM and Catia V5; the problem occurs when large assemblies must be loaded from Enovia VPM into Catia V5. This problem appears during or after positioning of the assembly components when Enovia VPM is closed accidentally; in this case the positions of the positioned parts in Catia V5 can't be saved in Enovia VPM.

In conclusion this tow macros it is very useful in practice because don't lost the time with repositioning pieces in the assembly interior.

After parts assembling using constraints from Assembly Design, with DMU Kinematics Simulator, it is made automatically the mechanism of this assembly (Fig. 9)

DMU Kinematics Simulator is an independent CAD product dedicated to simulating assembly motions. It addresses the design review environment of digital mock-ups (DMU) and can handle a wide range of products from consumer goods to very large automotive or aerospace projects as well as plants, ships and heavy machinery.

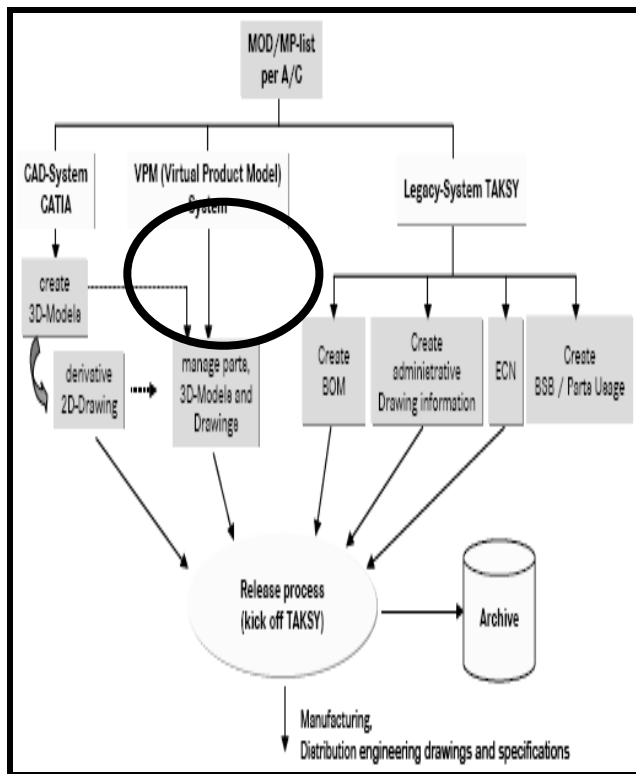


Fig. 8. Airbus Illustration of the Complete Process for Preparing Products.

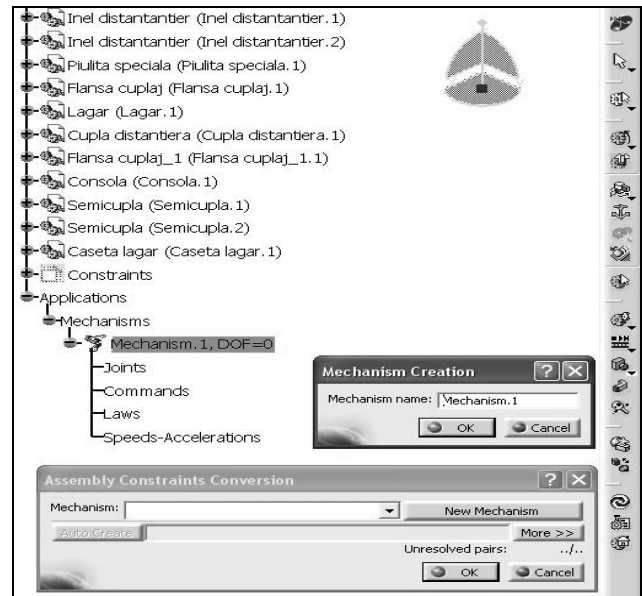


Fig. 9. Creating a mechanism in CATIA.

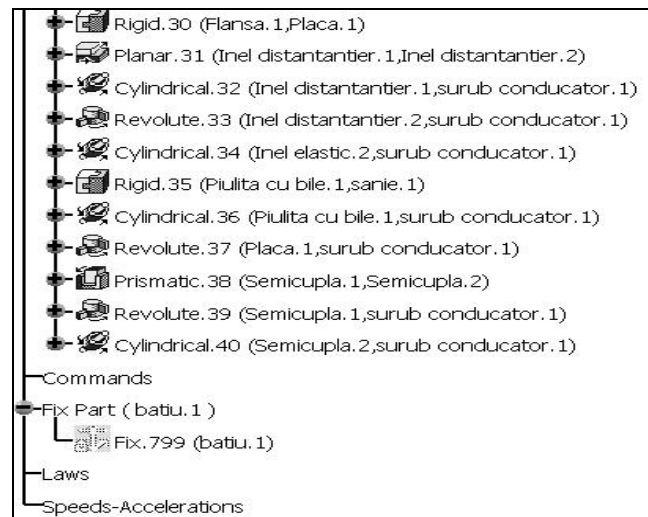


Fig. 10. Kinematics Joints in CATIA.

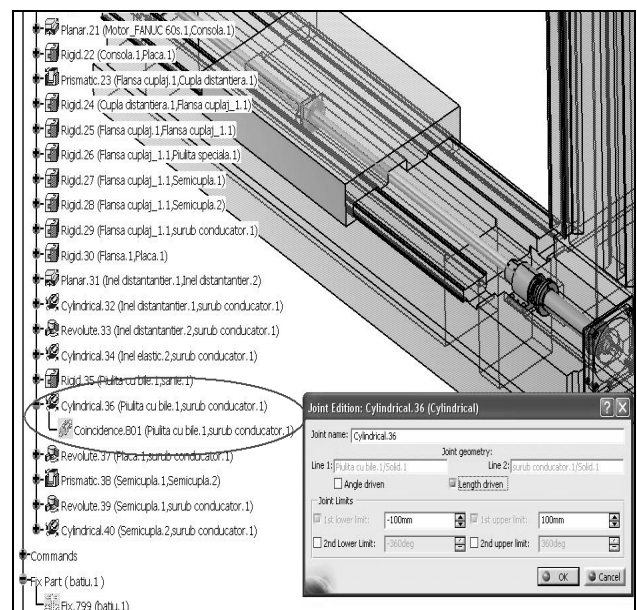


Fig. 11. The Joint Creation dialog box.

After the machine tool mechanism is created, can be simulated the parts manufacturing using NC Manufacturing from CATIA V5

CATIA NC Manufacturing Review 2 (NCG) offers basic numerical control (NC) capabilities such as tool path verification, material removal simulation, remaining material analysis, tool path edition and creation of shop floor documentation. It provides the infrastructure for all V5 NC programming products and allows numerical control (NC) programmers or machine operators to review V5 part operations. Furthermore, different manufacturing data such as APT files, CL files and ISO code can be imported and reviewed.

Picture 12 represents the tree structure from Catia with necessary resources and stages to manufacturing the part take as an example.

In most cases, numerical control (NC) programming supervisors or machine operators do not have a need for a full equipped CATIA V5 NC seat. CATIA NC Manufacturing Review 2 (NCG) allows them to review V5 part operations by browsing machining operations with all information on machining strategy, cutting tools, feeds and speeds, approach and retract macros and associated geometry.

CATIA NC Manufacturing Review 2 (NCG) enables to import any NC program available in APT file, CL file or ISO code and review it in a 3D graphic environment. You can replay the tool path replay and simulate the material removal. Modifications can be done by tool path editing. Output can be independent of import data format.

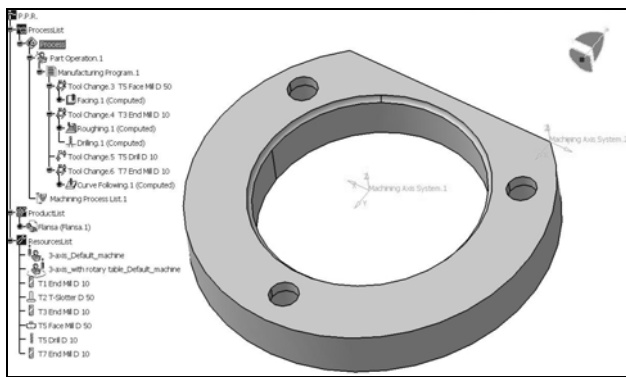


Fig. 12. Process Product Resources.

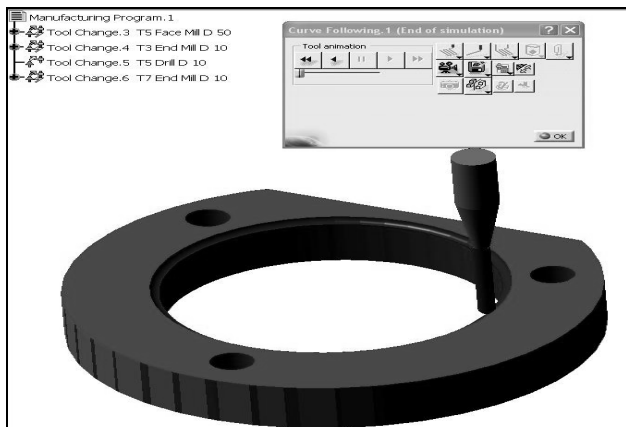


Fig. 13. Manufacturing program.

CATIA NC Manufacturing Verification 2 (NVG) offers advanced tool path verification capabilities for multi-axis positioning as well as for multi-axis machining. The accuracy of machined parts can be analyzed either by detection and display gouges and remaining material, by pick point analysis, or by measuring. Collisions between the tool or tool holder and part or fixtures are detected and graphically visualized.

The results of a material removal simulation can be stored in a reporting file.

3. CONCLUSIONS

Using main CATIA V5R18 modules it is recommended to evaluate mechanical assembly of machine or other equipments with high complexity parts.

Main joint conditions in work machine domain are surface contacts. The complex surfaces for the ball – screw mechanisms are imposed for the axis X, Y, Z and W of the milling and boring machine AF 105.

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