

VIRTUAL PROTOTYPE AND KINEMATICS MODELING FOR WAFER MANIPULATING ROBOTS

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Abstract: The paper presents advanced virtual prototyping modeling of a SCARA based IR, special dedicated for wafer manipulation in the electronics industry using Catia V5. Synthetic details about IR's parts, subassemblies and general assembly optimized design procedures as well as IR's mechanism kinematics and IR's overall functioning simulation are presented.

Key words: industrial robots, optimum design, virtual prototyping, wafer manipulating robot.

1. INTRODUCTION

The paper presents advanced virtual prototyping modeling of a SCARA based IR, (special dedicated for “wafer” manipulation in the electronics industry) using Catia V5. Synthetic details about IR's parts, subassemblies and general assembly optimized design and virtual prototypes achieving, as well as IR's mechanism kinematics and IR's overall functioning simulation performing are presented. The general assembly of the IR type, having 4 degrees of freedom (1T+3R), last 3 of them being synchronized in order to obtain a linear trajectory of the wafer manipulating end – effector is (Fig. 1),

To point out the linear/angular movements executed by the mobile elements of the SCARA robot, there have been used *Digital Mockup* menu and the commands related to *DMU Kinematics* sub-menu of *Catia V5* virtual prototyping environment.

DMU Kinematics Simulator is a CAD modulus for simulation of movements made by the partial / general assemblies of technical systems designed within *3D Catia modeler*. The Simulator may be used as a *3D independent modeler* for any virtual prototype or physical system digital simulation.

2. IR'S INDIVIDUAL PARTS DESIGN

For 3D modeling of IR's component parts, *Mechanical Design* menu and related *Part Design* sub-menu has been used. The most important commands of *Part Design* sub-menu are: *Sketcher* (used to design the 2D sketches) and *Sketch-Based Features* (the starting point

for modeling of any 3D part's design). Between these two commands *Exit workbench* command may be used. The *Sketcher* command includes two toolbars menus *Profile* and *Constraint* allowing, basically, the dimensioning and constraining of the 2D sketch that before it's using as a support for starting 3D modeling.

The *Constraint* option supplementary allows the visualization (step by step) of constraints imposed to the sketched profile, (warning the user about omission of some dimensions, over-constraints existence, inconsistency between some dimensions etc.) and validates the fulfillment of all constraints (Fig. 2).

As regards the possibilities of 3D sketch / model's visualization, *Catia* environment allows quick selection / modification of the viewpoint and visualization angle by “on-mouse” arrangement of commands *Pan*, *Zoom in*, *Zoom out*, *Rotate* (if a three buttons standard mouse is used). Herewith the selection of these commands from toolbars is not required (as the case of AutoCAD software package is). Thus the design time of 3D models may be considerably reduced.

For any 3D designed / imported model, by using *Apply material* command, the user can choose from a database or respectively define itself, the characteristics of model's material. (Fig. 3a). From the active window opened using this command, a series of constructive-geometrical characteristics of 3D model (such as mass, volume, inertia moment and so on) can also be obtained (Fig. 3b). For a complete determination and verification of 3D model's constructive-/ geometrical characteristics *Drawing* menu and *Wizard* command (Fig. 4) may be

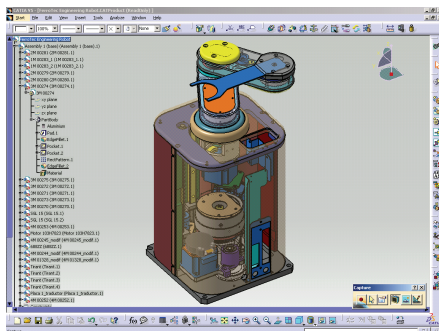


Fig. 1. General assembly of a wafer manipulating robot.

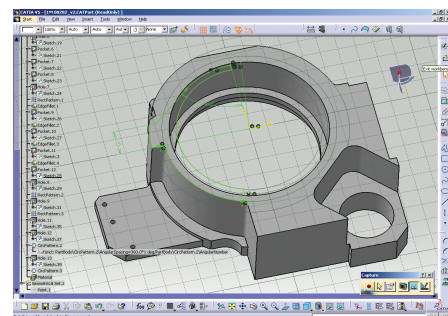


Fig. 2. Visualization example of 2 D profile closing using „Exit workbench” command.

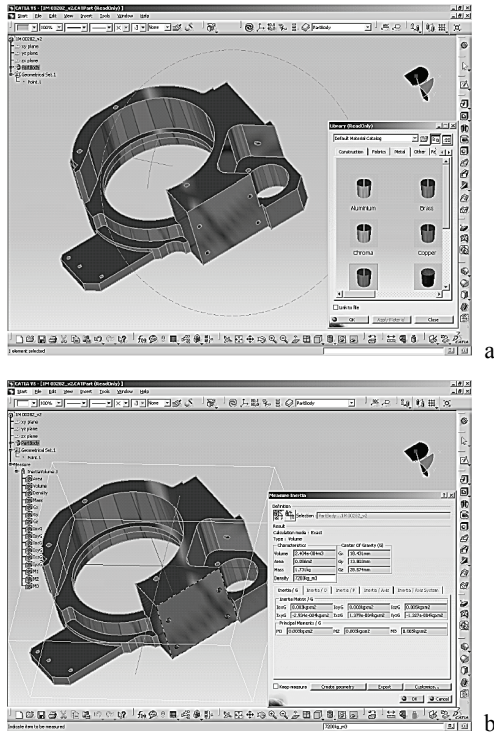


Fig. 3. Using Apply material command: a) material selection facilities for quick visualization of 3D models; b) material's characteristics visualization.

complementary used (for selecting specific visualization angle – Fig. 4, right down corner) or crosschecking 3D models and part's manufacturing drawing, as well as *Split* command (allowing model's sectioning by different planes as well as choosing orthogonal or axonometric viewing of the part (Fig. 5).

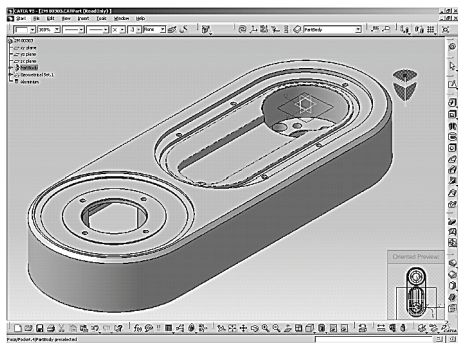


Fig. 4. Completion and verification of 3D model.

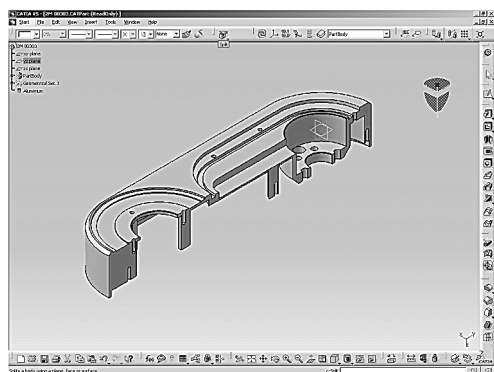


Fig. 5. 3D model sectioning.

The manufacturing / assembly drawings dimensioning is easy to achieve too. For this purpose *Catia* environment offers three possibilities: automatic generation of dimensions (*Generating Dimensions* command), free dimensioning (by the user) or appealing to a combination of above mentioned two methods. All modifications made on constructive shape / dimensions characteristic to a 3D model shall be found on the manufacturing drawing of the modified part as well as on the assembly in which this is integrated. Their updating is automatically made in parallel to modifications bought on 3D model.

3. ASSEMBLY'S DESIGN. JOINT'S AND BASIC MECHANISM'S KINEMATIC MODELLING. SIMULATION OF ROBOT FUNCTIONING

To design the partial assemblies / general assembly of a product in *Catia* environment, the *Assembly Design* sub-menu of the same *Mechanical Design* menu is used.

Besides of 3D models and components made by the users, in order to achieve IR's sub-assemblies (Fig. 7), partial assemblies (Fig. 8, Fig. 9 and Fig. 10) as well as IR's general assembly (Fig. 1), it also may be used the

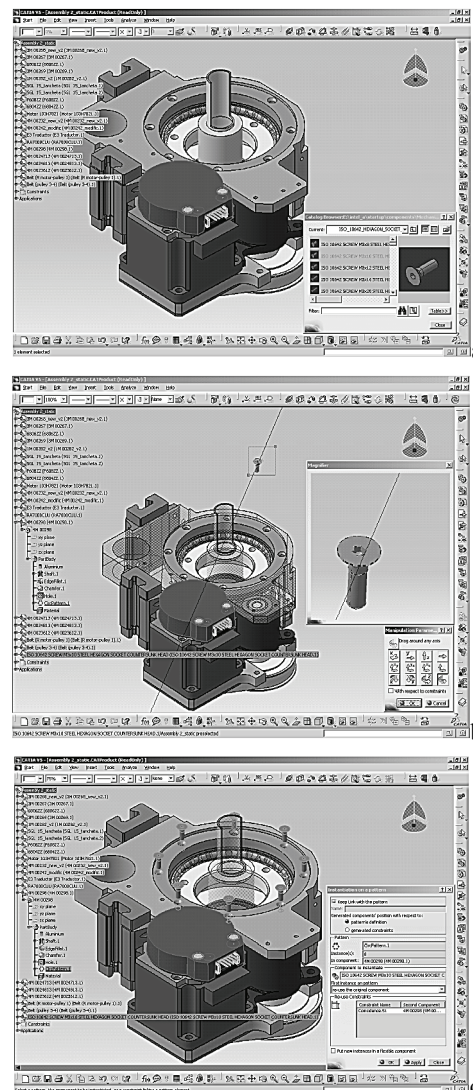


Fig. 6. a) Selection of one part using CAD 3D catalogue; b) manipulation and Zoom commands; c) Insertion of all parts.

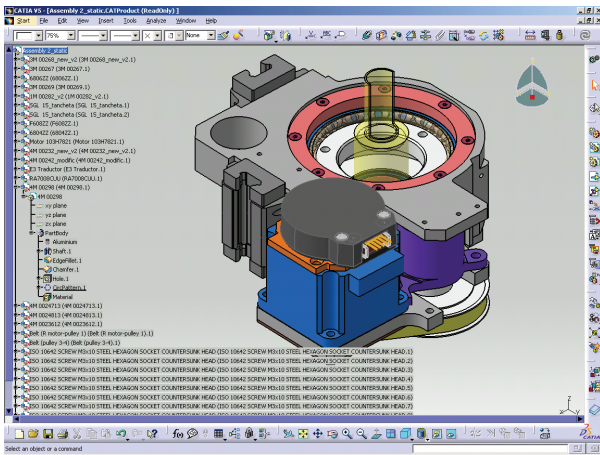


Fig. 7. Designed partial assembly.

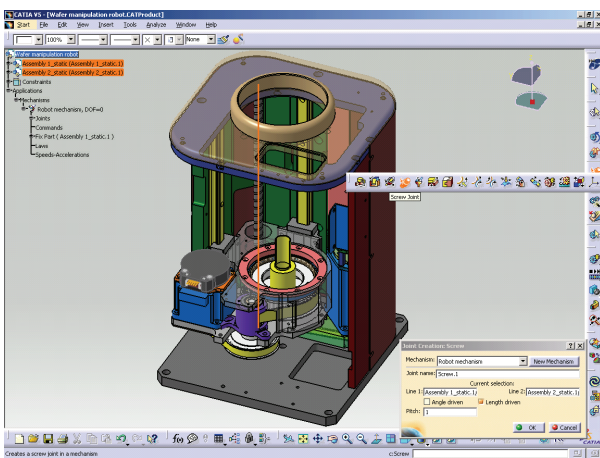


Fig. 8. Using Screw Joint command.

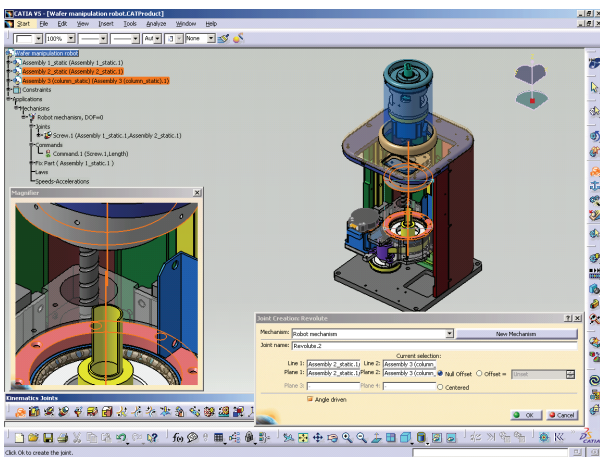


Fig. 9. Using Revolute Joint command.

3D standardized part's catalogues predefined in *Catia*, as Fig. 6 (a, b, c) shows (highlighting succession of operation that should be made for selection / insertion of some screws for reciprocally assembling two distinct parts). It may be observed that in order to insert all six screws it is required the fulfillment of the selection procedure for only a single screw (Fig. 6a). Other assembly elements can be automatically inserted, subsequently to this procedure by using a single command too (Fig. 6c).

In order for reaching an adequate / suitable position of different parts before assembling them, *Catia* envi-

ronment offer specific facilities, integrated in *Move* toolbar. Thus, by using *Manipulation* command (Fig. 6b) is it allowed to translate a part on X, Y, Z axis's direction / XY, YZ, ZX plane's directions, or (when one part's edges has been previously selected) following a direction parallel to one part's side / edge, as well as to rotate the part around each X, Y, Z axis, or around own part's axis (in case of a cylindrical part type).

For a better visualization of part's characteristics included in sub-assemblies / partially assemblies, or as well much easy assembling them, *Catia* environment allows changing part's color / selecting their transparency level (Fig. 6b) or temporary hiding them as well as preliminary visualize particular sections of an assembly by means of „zoom in” window from *Magnifier* option (Fig. 6b).

When all parts, sub-assemblies and assemblies are fulfilled, the general assembly of the robot (Fig. 1) may be achieved and kinematical modeling performed.

For the modeling of the joints corresponding to the first two DOF of IR effector's positioning system (one translation and one rotation), there have been used a balls – screw – nut joint type to generate IR's column vertical translation and a rotational joint for the rotation of robot's column. To connect these two mechanisms on the driving motors, two timing belt driven mechanisms have been used.

In order to define the functional parameters of above mentioned specific mechanisms and respectively perform kinematics simulation of IR's positioning system, 1-st IR's DOF has been modeled as a translation joint, by mean of *Screw Joint* command. Thus, in Fig. 8 it may be observed the mobile assembly on IR's Z axis (1-st IR's NC translation axis), including too the driving system designed to obtain the rotation movement of the column (2-nd IR's NC axis) as well some other elements of these two partially assemblies. The right side of the picture includes the toolbars as well the dialog window that shall be opened when using *Screw Joint* command.

The rotation movement of the column (2-nd IR's NC axis) joint has been defined between the two partially assemblies. In the left side of the figure, it can be observed the toolbar (including the commands that may be used to define the joints related to this specific IR's DOF) and respectively, in the right down – corner, the dialog window that is opening when this command is used (for defining specific cinematic parameters of this joint).

To define the 3-rd IR's DOF (end-effector radial translation in horizontal plane), combined movements need to be modeled as three reciprocally synchronized rotations (column / IR arm's 1-st link rotation, IR arm's 2-nd link rotation and end-effector rotation). For this purpose it has been used *Revolute Joint* command and adequate defining of the functional parameters for each revolute joint (Fig. 10).

After characterization of each joint type, the user receives a screen message which inform that everything is OK and the mechanism can be simulated or respectively that he needs to complete characterization of some joint's parameters / continue by other commands.

For each above mentioned mechanism, the user can select a series of cinematic characteristics such are number

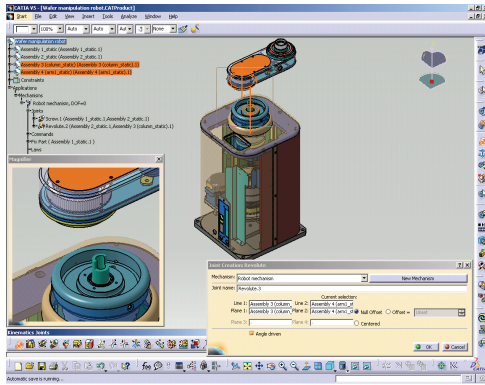


Fig. 10. Using Revolute Joint command.

of defined joints and commands, number of DOF, which is the fixed / moving element within partial / general assembly etc.

Depending on errors made during mechanism's parameterization (if the case) the user may be accordingly informed that the mechanism is over-constrained or the command for defining the revolute joint will have no effect, thus the cinematic simulation of the mechanism / sub-assembly is possible or not.

However, if the command / commands for the simulation of joints movement has / have been correctly defined and the mechanism's simulation can be started, the user shall have afterwards to his disposal the option to modify the cinematic parameters of each / all movement(s) (as may be establishing the distance for end-effector linear movement on radial translation axis, or adjust the rotation angle on a rotation axis, or as well adjust the number of steps to be performed in the cinematic simulation).

Thus, assuming that all types of joints for each kind of mechanisms included in IR's general assembly have been already defined as above, the user shall have following two options for starting simulation and visualize it:

- By using **Simulation With Commands** option, allowing successive cinematic simulation of each previously defined mechanisms (Fig. 11);
- By using **Simulation** option, allowing the cinematic simulation of each / overall mechanisms defined within an assembly by either simultaneous or separately motions (generated by all / each mechanisms).

Further to last of above-mentioned commands, it may be also performed works necessities to elaborate a video presentation (animated movie) of the cinematic simulation,

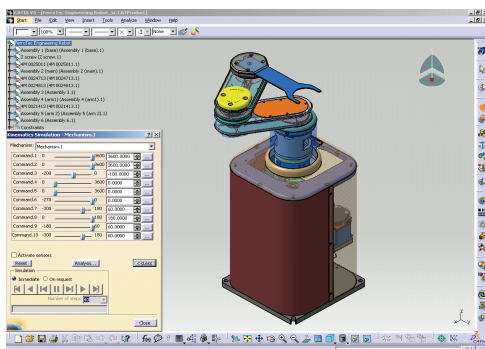


Fig. 11. Simulation With Commands option.

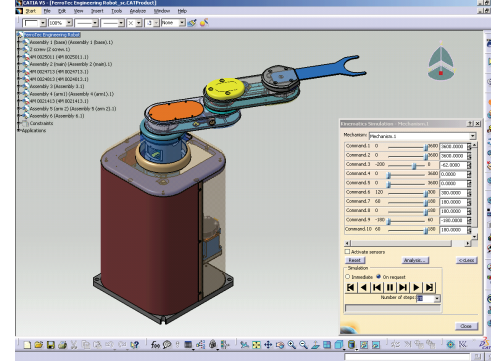
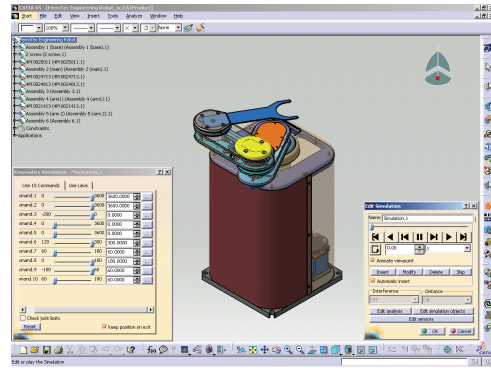


Fig. 12. Simulation option.

as Fig. 12 is showing, for two different arrangements of the mobile elements of the robot during the simultaneous cinematic simulation of four of the five IR's main mechanisms.

As result, finally, by appealing to just above mentioned commands, the user will got the possibility to effective visualize IR's individual / overall movement(s), or even study eventual collisions between IR's mobile elements and the rest of some other sub-system(s) included in a common (application's) assembly.

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