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VIRTUAL PROTOTYPE OF INDUSTRIAL ROBOTS AND FLEXIBLE MANUFACTURING CELLS USING CATIA

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Abstract: *The paper presents advanced virtual prototyping modeling using Catia V5 of a flexible manufacturing cell for high speed machining, including a 5 DOF gantry robot, a linear axis part's positioning system and a set of automatic closure doors delimitating a "protected area". The 5 DOF gantry industrial robot is special dedicated for high speed machining of dies, as well as any other kind of similar parts including 3D complex surfaces. Synthetic details about IR's parts, subassemblies and general assembly optimized design procedures as well as IR's kinematics and IR's overall functioning simulation are presented.*
Key words: *industrial robots, optimum design, virtual prototyping, high speed machining, dies manufacturing*

1. INTRODUCTION

The paper presents advanced virtual prototyping modeling using Catia V5 of a flexible manufacturing cell (FMC) for high speed machining, including a 5 DOF gantry industrial robot (IR), a linear axis part's positioning system and a set of automatic closure doors delimitating a "protected area". The 5 DOF gantry robot is special dedicated for high speed machining of dies, as well as any other kind of similar parts including 3D complex surfaces. Synthetic details about IR's parts, subassemblies and general assembly optimized design and virtual prototypes achieving, as well as IR's kinematics and IR's / FMC's overall functioning simulation performing are presented.

The general assembly of the gantry type IR, includes specific partial assemblies for all 5 DOF (3T+2R) and IR's end-effector (high speed machining modular spindle), first three DOF being used for IR's end-effector positioning on X / Y / Z axis and the last 2 of them for roll – pitch end-effector orientation (Fig.1), [1].

After individually defining of each IR's specific parts and partially assemblies, in the cinematic modeling, all above 5 DOF have been individually defined, reciprocally constrained and specific motion laws have been defined to allow end-effector's trajectory generation and servo-assisted end-effector's orientation simulation [2]. Complementary works for fully synchronization of IR's overall (5 DOF) kinematics with the auxiliary linear positioning axis and the automatic closure doors motions have been performed too in order to achieve overall FMC's functioning simulation. To point out the linear/angular movements executed by the mobile elements of the gantry 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 [3].

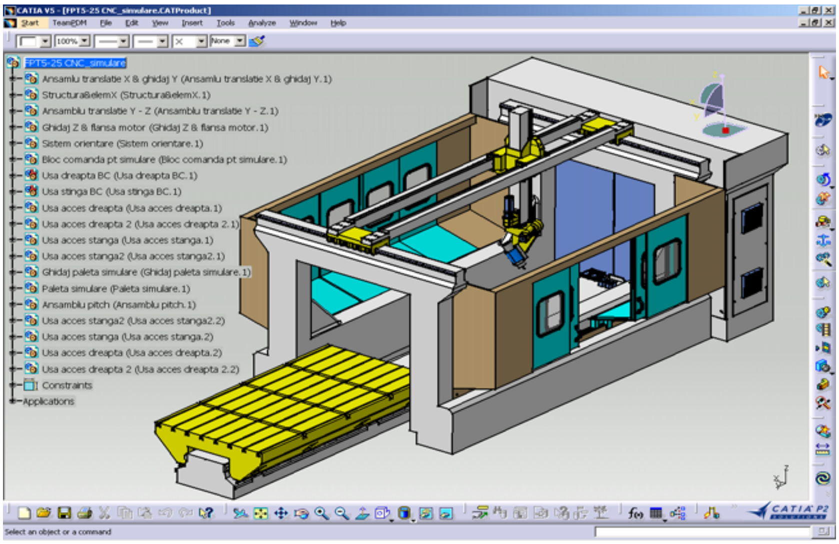


Fig. 1. *General assembly of the FMC and gantry IR*

2. IR'S INDIVIDUAL PARTS DESIGN

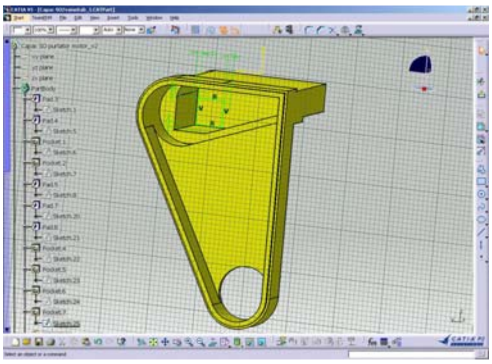
For 3D modeling of IR's component parts, ***Mechanical Design*** menu and related ***Part Design*** sub-menu has been used. The most important commands of ***Part Design*** sub-menu are: ***Sketcher*** (used to design the 2D sketches) and ***Sketch-Based Features*** (the starting point for modeling of 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, inconsistence between some dimensions etc.) and validates the fulfillment of all constraints (Fig.2a). 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 (Fig. 2b).

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. 2c). 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. 2d). For a complete determination and verification of 3D model’s constructive-/ geometrical characteristics ***Drawing*** menu and ***Wizard*** command (Fig. 2e) may be complementary used (for selecting specific visualization angle - Fig. 2e, 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. 2f).

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.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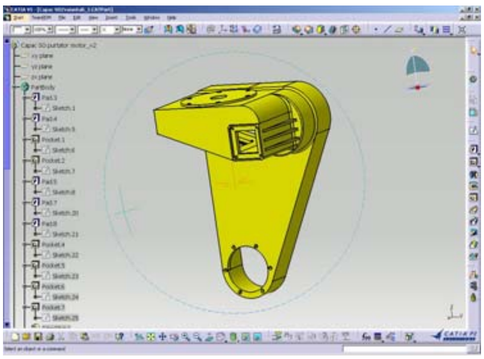
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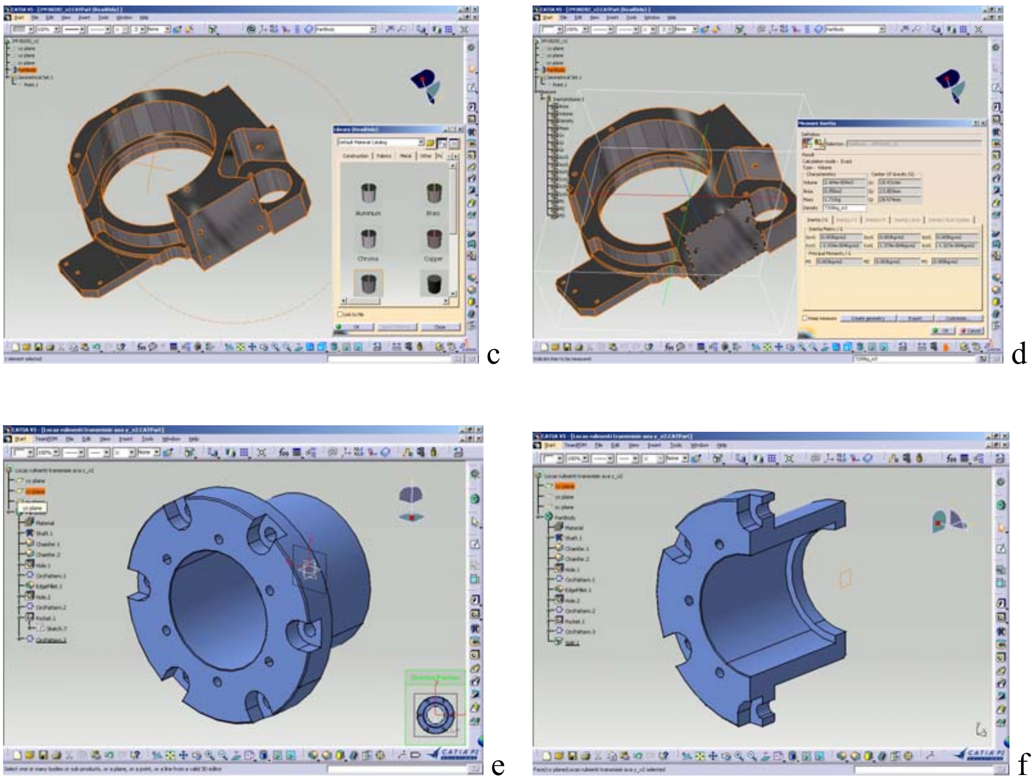


Fig. 2. Using “*Part Design” sub-menu: a) Visualization example of 2 D profile closing using „Exit workbench” command; b) “Pan, “Zoom in”, “Zoom out”, “Rotate commands”; c), d)“Apply material” command; e), f) “Wizard” and “Split” commands*

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 / partial assemblies (Fig. 4a, b, c, d, e, f, g, h, i) as well as IR’s / FMC’s general assembly (Fig. 1), it also may be used the 3D standardized part’s catalogues predefined in ***Catia,*** as Fig.3a, b 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. 3a). Other assembly elements can be automatically inserted subsequently to this procedure by using a single command too (Fig.3b).

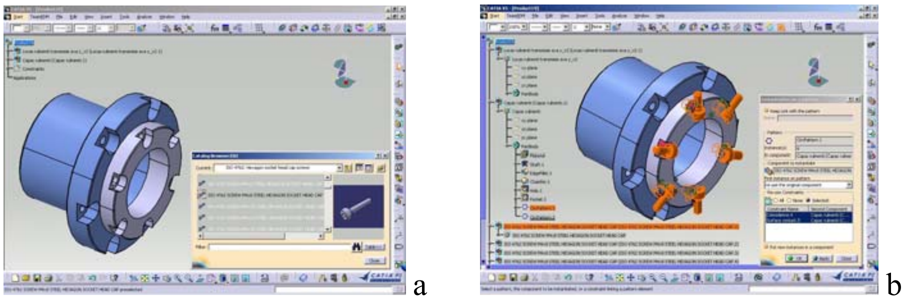


Fig. 3. *a) Selection of one part using CAD 3D catalogue;*
b) Manipulation, Zoom and Insertion of all parts.

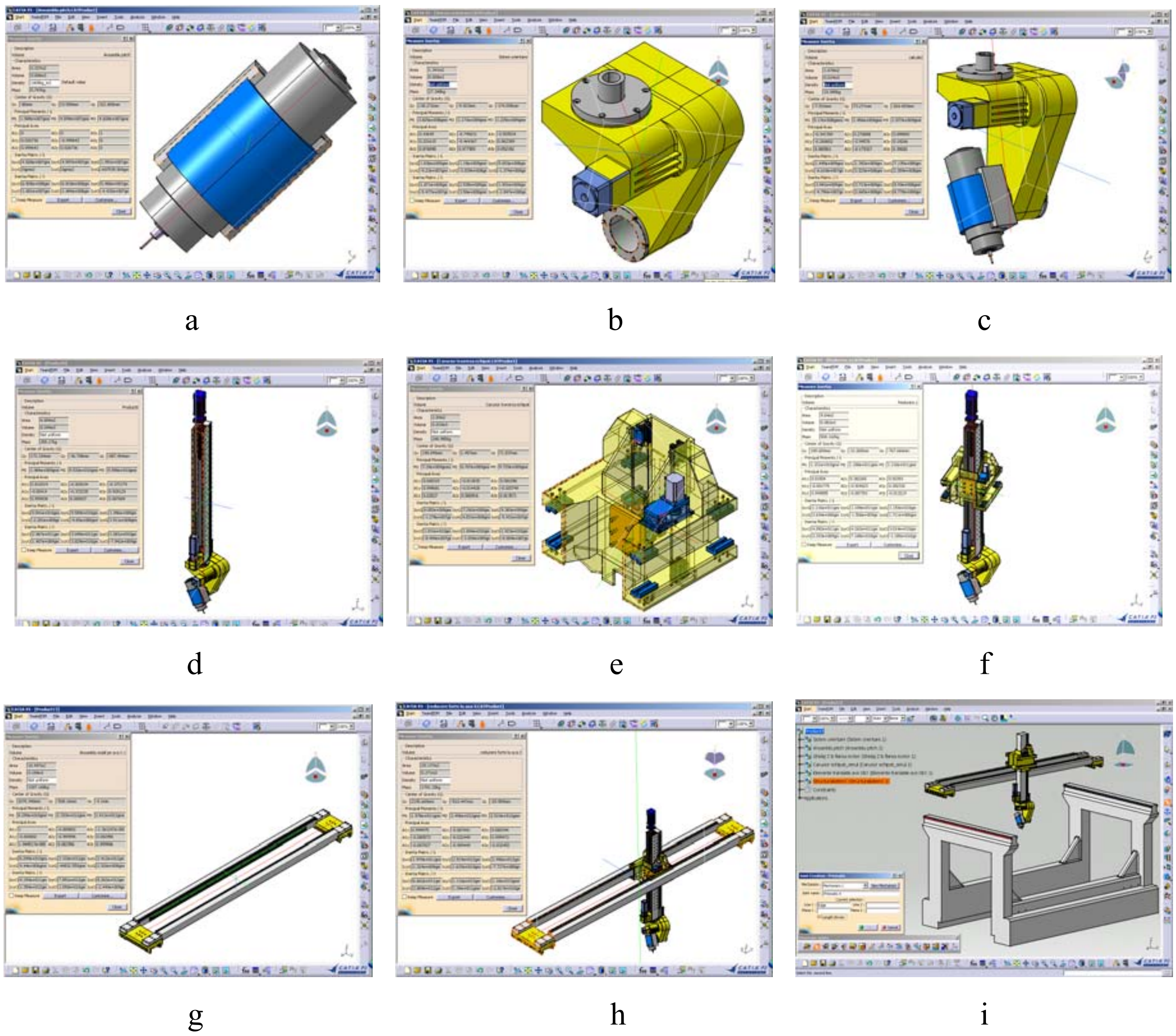
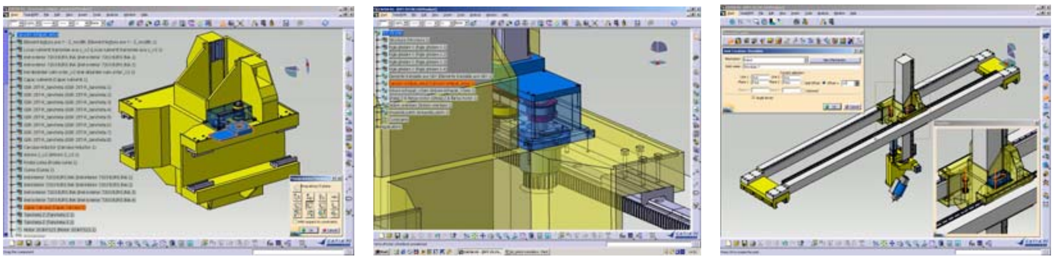


Fig. 4. *Designed partial assembly: a) IR's end-effector; b), c) pitch – roll orientation axis; d), e), f), g), h), i) Z, Y, X positioning axis*

In order for reaching an adequate / suitable position of different parts before assembling them, ***Catia*** environment offer specific facilities, integrated in ***Move*** toolbar. Thus, by using

Manipulation command (Fig. 5a) 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).

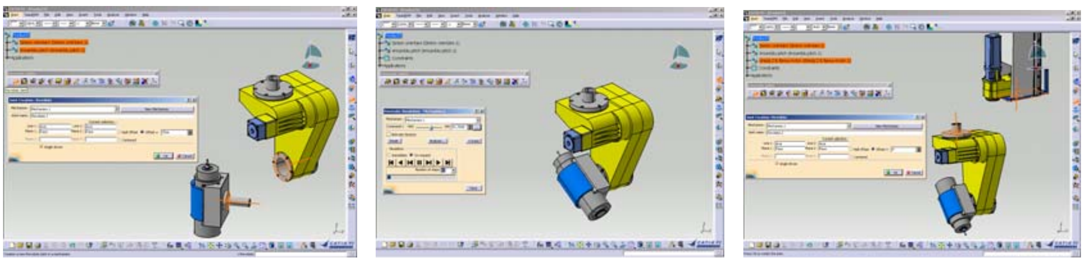
For a better visualization of part's characteristics included in sub-assemblies / partially assemblies, or as well much easy assembling them, ***Catia*** environment allows changing part's color / selecting their transparency level (Fig. 5b) or temporary hiding them as well as preliminary visualize particular sections of an assembly by means of ***„zoom in"*** window from ***Magnifier*** option (Fig. 5c).



a b c

Fig. 5. *a) "Manipulation" command; b) Selecting part's transparency level;*
c) Visualizing particular sections of an assembly by using "Magnifier" option

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 last two DOF of IR's end-effector orientation (pitch-roll) there have been used rotational joints (Fig. 6a, b, c, d), and respectively, for the first three DOF of IR end-effector's positioning system (X, Y, Z translation), there have been used translational joints (Fig. 6a, b, c, d).



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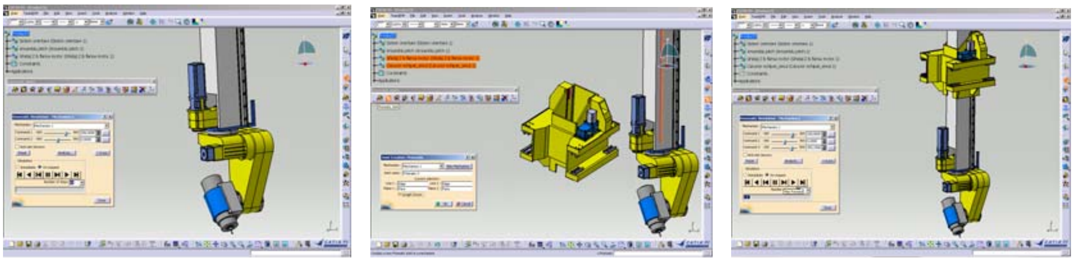

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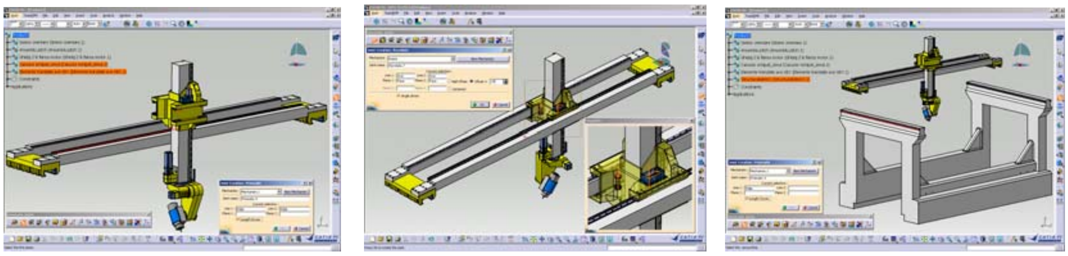

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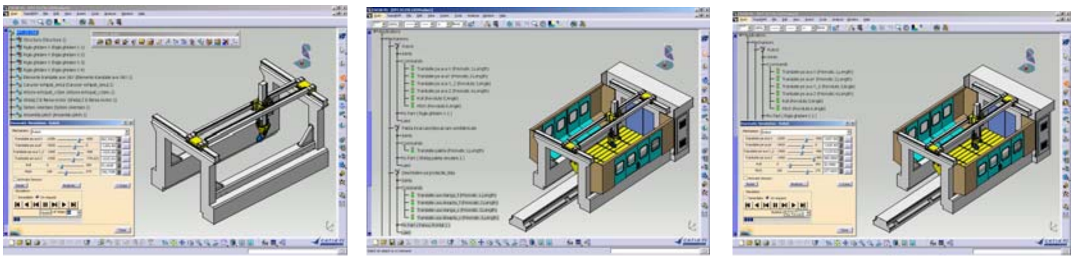

j k l

Fig. 6. *IR's Cinematic modeling: a), b), c), d) Using "Revolute Joint" command; e), f), g), h), i), j), k), l), Using "Prismatic Joint" command*

For defining specific cinematic parameters of all joints, both, rotational movements of 4-th and 5-th IR's NC axis as well as translational movements of 3-th, 2-nd and 1-st IR's NC axis have been defined a series of cinematic characteristics such are: number of defined joints and commands, number of DOF, which is the fixed / moving element within partial / general assembly, etc. Taking into account all above a final check of complete characterization for all joint's parameters before the cinematic simulation of each individual joint / overall IR's virtual prototype has been performed.

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, adjust the rotation angle on a rotation axis, or as well adjust the number of steps to be performed in the cinematic simulation etc.

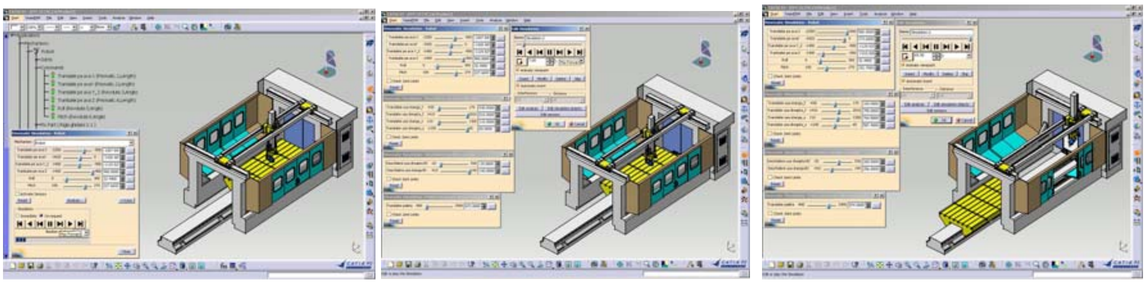

Fig. 7 Overal *IR’s /FMC’s cinematic modeling:*

a) Using “Simulation With Commands” command; b) Using “Simulation” command

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: ***Simulation With Commands*** option, allowing successive cinematic simulation of each previously defined mechanisms (Fig. 7a); and ***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 (Fig. 7b).

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