

SIEMENS SOLUTIONS APPLIED TO OBTAIN MECHANICAL ASSEMBLIES IN AGRICULTURE

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ABSTRACT

The paper provides a brief description of some common aspects to obtain assemblies for use in agriculture by applying SIEMENS PLM NX 7.5 design solution launched in June 2016. In the paper are also exemplified peculiarities of assemblies opening, details about Assembly Navigator and a brief description is provided on how to apply the methods Bottom-Up and Top-Down for modelling three-dimensional assemblies with NX 7.5.

KEYWORDS: modelling, assembly, CAD, NX 7.5

1. GENERAL REMARKS ON 3D ASSEMBLIES

To give some explanations about 3D assemblies made with SIEMENS NX 7.5, the starting point is that the essential particularity of this design software is the use of a single file with the extension *.prt which contains all the information needed to be transmitted among its users. The vast majority of designed software have types of file extensions for each working sector, such as: 3D model for a single item, 3D model for the assembly, 2D documentation for execution, 3D model for welding or 3D model for pieces of thin plates. It should be noted that the file*.prt includes all these situations treated distinctly from other design software and may contain additional information about simulation and manufacturing. In other words, an assembly contains pieces of distinct item type or subassemblies which in turn are made up of other distinct items noting that all these components are taken and analyzed with their entire collection of physical and mechanical properties and also with the entire set of links among them, such as constraints associated with an assembly. Constraints, parametric in nature, preserve the unity of the assembly, according to the purpose of modelling. Any modification of the constituent parts or their relationships will result in automatic updating of the whole assembly, just as when modelling individual parts. Although an assembly may contain both geometry and 3D components with their own

geometry, it is recommended [1] to separate modelling from building assemblies and possibly to use workforce also separately.

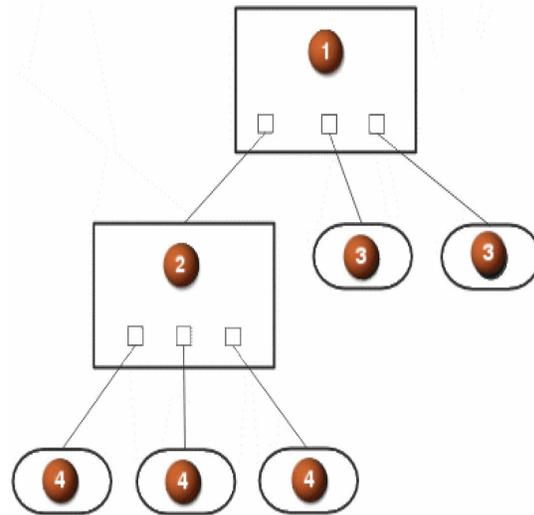


Fig. 1 Example of assembly scheme

Figure 1 schematically illustrates an assembly made up of distinct items like components labelled "3" and a subset labelled "2" which in turn contains its individual parts marked with "4".

2. PARTICULARS OF OPENING ASSEMBLIES MADE WITH NX 7.5

To start a new assembly-type file, the command "New" is launched and the file Assembly is selected from the window shown in Fig.2.

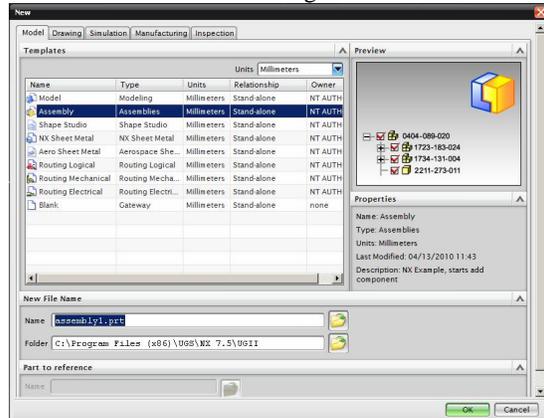


Fig. 2 „New” window [2]

Either immediately after opening the file or before selecting the assembly file, the rules for loading the components should be set, by ordering "Assembly Load Options" to be found in the "File" menu and its path for launching is shown in Figure 3.

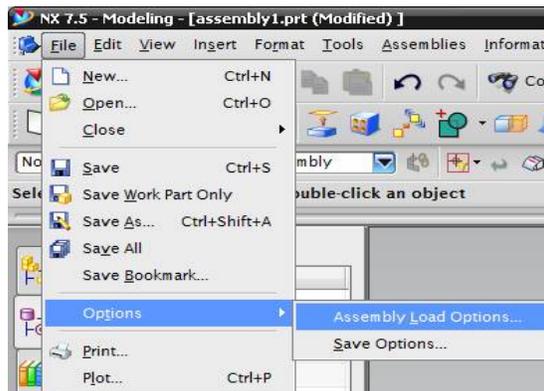


Fig. 3 Localization of „Assembly Load Options” command

If you open an assembly-type file generated at an earlier time, the user has the possibility to push the "Options" button in the opening and thus reach the stage of establishing the rules for loading the component parts. Figure 4 shows the window corresponding to command "Assembly Load Options" in which the following 6 setting zone can be seen: „Part version”, „Scope”, Load Behaviour”, „Reference Sets”, „Bookmark Restore Options” and „Saved Load Options” [3].

2.1. „Part Version” zone

This area contains the mode of selecting three variants of searching the parts to be loaded in the assembly- type file, namely:

- "As Saved" - a situation in which parts are loaded directly from the directory where they were saved;
- "From Folder" - pieces are loaded from the directory in which the parent-assembly is to be found. It is the default variant to open the window.

"From Search Folders" - a situation in which the parts are loaded from directories in a list of previous searches.

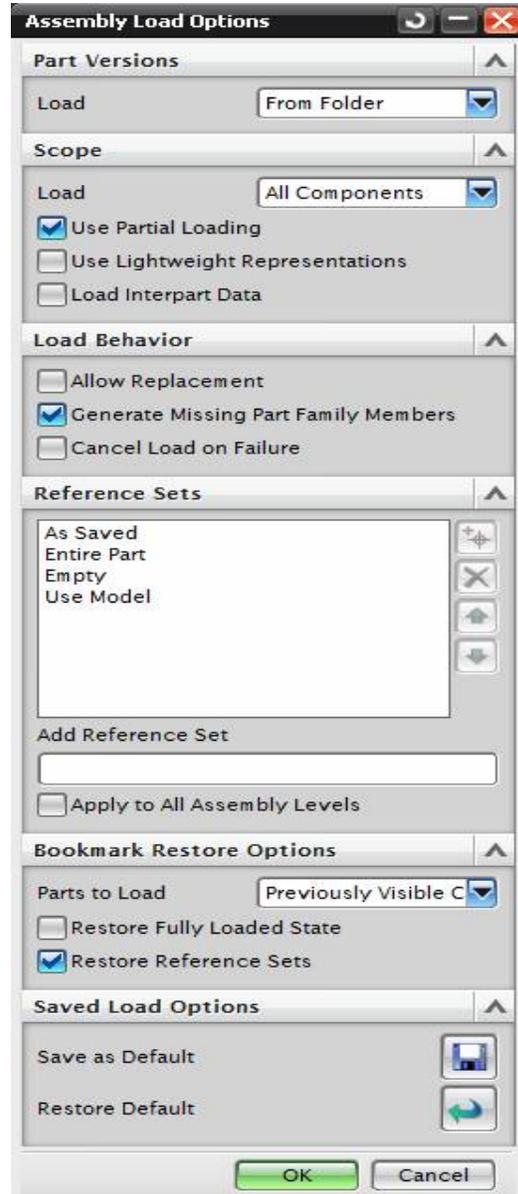


Fig. 4 „Assembly Load Options” command

2.2. „Scope” Zone

The "Scope" zone in the dialog "Assembly Load Options" window allows configuration control of the

assembly and loading characteristics in the following variants:

- Total loading when nothing is ticked in this area and consequently all data are loaded into the system memory.
- Partial loading of the parts of the assembly when loading only data required for viewing the part, and this loading is done directly in the memory.
- Loading of "Load Interpart Data" type when the software finds and loads only associative relationships among components, even if the parts would be left unloaded by other rules such as "Use Partial Loading".

The list of options of the area "Scope" contains:

- "All Components" – when all parts are loaded as individual parts;
- „ Structure Only" – when only the structure is loaded , i.e. the assembly is loaded, not the component parts;
- „As Saved" - when are loaded only the parts / components that were opened at the last save of the assembly.
- „Re-evaluate Last Component Group" –only the assembly will be loaded with the group of components used at the last save of the assembly.
- „Specify Component Group"- when the assembly is loaded only with the group of components.

2.3. „Load Behaviour" Zone

This area shows how to select one or more options from the three settings below:

- "Allow Replacement" –where if there is a difference between the file name and its identifier, the user receives some type of warning but the component is still loaded.
- „Generate Missing Part Family Members"- when opening the assembly, NX cannot find a particular component and it will search and load a newer version of that family of parts the part belongs to.
- „Cancel Load on Failure" – if NX fails to find a certain component, then the entire loading operation is cancelled.

2.4. „Reference Set" Zone

The "Reference Sets" zone is used in determining the order in which reference sets should be searched in the list. Figure 4 shows the default version that contains these four references: "As

Saved," "Entire Part", "Empty", "Use Model". For example, the set "Use Model" will be used as default only when the component brought into the assembly contains only the solid geometry without sketching, without quotas or reference elements. Sometimes it is desired, instead of a whole piece, to be loaded only a schematic representation, but this scheme was included previously in the current set of references.

2.5. Zones „Bookmark Restore Options" and „Saved Load Options"

These zones refer to restoration options of the lists of parts that belong to an assembly but also to the modalities of saving assemblies for their next openings.

3. DETAILS OF „ASSEMBLY NAVIGATOR"

The window "Assembly Navigator" in Fig. 5 can be viewed by pressing the appropriate button, in an assembly -type file, in toolbar "Resource" above the label "Part Navigator". Each component of an assembly appears as a node in the assembly tree structure.

Selecting a node is the same as when selecting the corresponding components in the graphics window. Each node contains a check box, an icon, part name and additional columns.

If the part is an assembly or a subassembly, an additional box will appear for deployment / collapse.

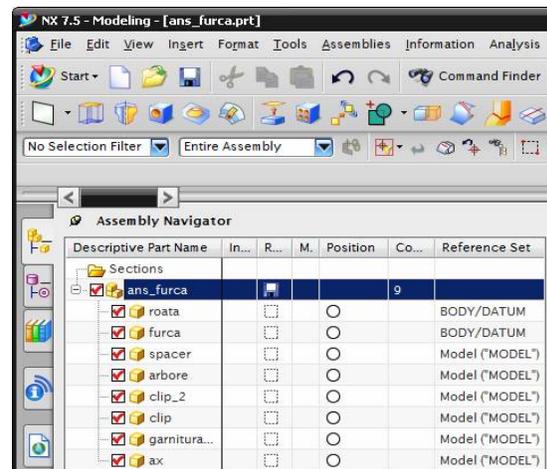


Fig. 5 „Assembly Navigator" window

The window "Assembly Navigator" contains a series of icons and check boxes of multiple meanings, among which: assembly or subassembly that is or is not under working, loaded or unloaded part, hidden subassembly or part, expanded subassembly or assembly, etc. Moreover, for greater efficiency of work within the assembly-type, the application "Assembly" should be activated. This application can be found in the "Start" list of the "Standard" toolbar , as shown in Figure 6 which contains a hydraulic

cylinder assembly of an agriculture equipment for digging pits.

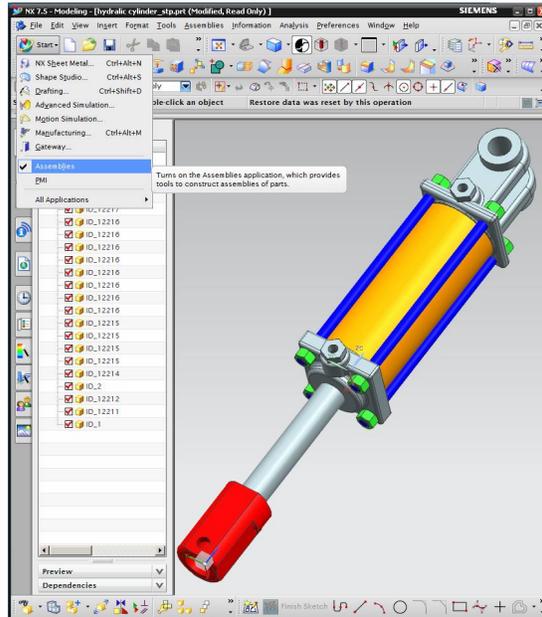


Fig. 6 „Assembly Navigator” window

4. METHODS OF CREATING MECHANICAL ASSEMBLIES IN AGRICULTURE

At this point we know two methods of modelling 3D assemblies, namely bottom up modelling and top-down modelling, each having advantages and disadvantages but also a well defined scope of application . The two methods are shown in Figure 7.

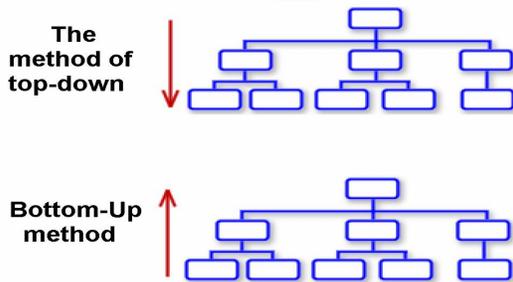


Fig. 7 Methods of modelling assemblies

Thus, if it is desired to create the 3D model starting from a malfunctioning assembly in order to be made again to the same output parameters, the bottom up modelling will always be used, ie each item separately will be 3D modelled and in the next stage to insert each piece into "Assembly" file imposing the 3D restrictions of relative contact . In situations of actual design, when the assembly of the object to be designed does not physically exist and the 3D model file is the result of the imagination and the design of the work team, the unanimous

recommendation is to apply the top-down method or maybe the mixed method on portions. Figure 8 shows the result of the Top-Down modelling of some pit digging equipment for agriculture.

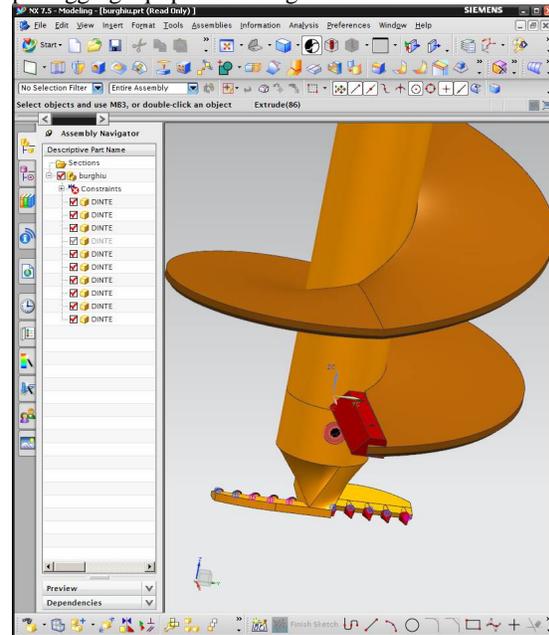


Fig. 8 Pit digging drill for agriculture

5. CONCLUSIONS

From the situation presented above, namely modelling of mechanical components that are part of the digging holes equipment for planting trees, it is concluded that both 3D modelling methods are perfectly suitable for the design of mechanical components used in agriculture.

The advantages of the second method are:

- Modelling parts is carried out with the assembly displayed.
- The possibility of associative copy of different geometries between different parts.
- The change in the source part geometry is propagated instantaneously to the destination part.

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