

NX I-deas Master Assembly

Benefits

- Manage large and complex products with multiple configurations
- Design in the context of the product assembly
- Reduce costs of physical prototypes by accurately representing product digitally
- Improve overall product quality by ensuring product fit and function
- Use JT™ data format parts to design with data from other CAD products in the context of the assembly

Features

- Bottom-up or top-down assembly hierarchy definition
- Assembly hierarchy navigation tools for large assembly management
- Pruning capabilities to limit screen just to necessary component geometry, with other non-essential components left in the data management environment
- Configuration management tools to manipulate orientations and other assembly information such as hide/suppress status and orientation changes

Summary

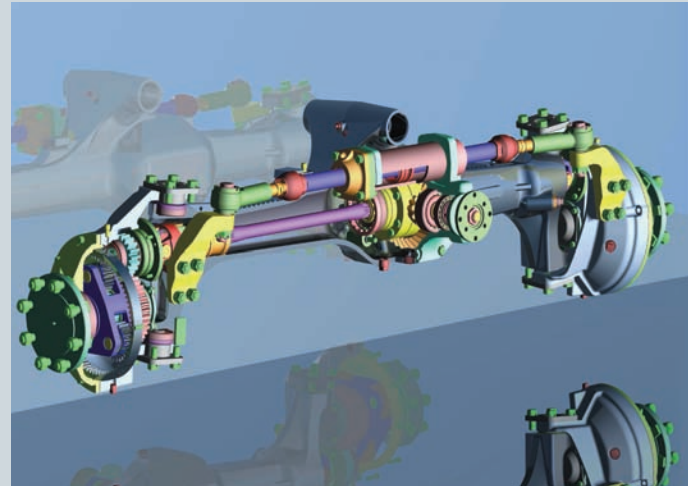
NX™ I-deas™ Master Assembly is an add-on module within NX I-deas software that brings the product together in a collection of assemblies and subassemblies. Assemblies defined in Master Assembly leverage parts created using NX I-deas Master Modeler and NX I-deas Master Surfacing software, and capabilities from these other applications can also be used to design while in the context of the product assembly. By easily simulating the behavior of product design digitally, Master Assembly helps to reduce the costs of physical prototypes and ensures higher product quality through better fit and function.

Capabilities for assembly, design and evaluation

Master Assembly allows you to work in a multi-user environment to layout, design and manage large mechanical assemblies. Master Assembly provides tools to capture and manage design intent at the assembly level, including interpart relationships, constraints that drive orientations and configurations that manage different product possibilities. It shortens design time and improves design quality by simplifying packaging and interference studies.

Model large assemblies

Master Assembly provides a comprehensive set of interactive tools for creating and manipulating large mechanical assemblies. Parts are positioned relative to other parts using intuitive constraints and dimensions which define design intent. Users can dimension and



Master Assembly allows you to create a complete electronic mockup of your product design. Intuitive positioning commands and interpart relationships allow you to capture design intent and make design configuration changes easily.

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Features *continued*

- Assembly constraints to capture and represent product level design intent
- Interpart geometrical relationships defined using associative copy of geometry across parts
- Interference and clearance checking for both static positions as well as across a range of motion stored in a sequence of configurations

constrain parts relative to one another with natural and intuitive methods that closely represent the physical assembly process. Design changes to one part ripple through the entire assembly, dynamically updating part and subassembly positions.

Master Assembly modeling features include:

- Large model size allows you to create a complete electronic product mockup
- Flexible methods for part assembly orientation help you modify the orientation of parts in subassemblies

- Users can create a pattern of assembly instances associated to a pattern within a part. For example, a bolt, washer and nut can be tied to a bolt-hole pattern. The number and positions of the instances will automatically update with changes to the hole pattern
- Users can employ Assembly Reflect to save time creating symmetrical assemblies

Pruning broadens the NX I-deas toolkit for managing large assemblies. By reducing or eliminating the data moving across the network, performance is increased and model file size is reduced:

- Retrieve and manipulate product structure without waiting to retrieve all of the associated parts from a library
- Model individual parts in the context of a large assembly
- Build overall product structures or bills of material (BOM) without any parts
- Check fit and function of one or more components that you are responsible for in the context of the entire product structure
- Reference components from a variety of assemblies to preclude interference problems
- Get the components you need, with the same orientations they have within the context of the assembly that captures the product's design intent
- Retrieve only those parts within a specified distance of a selected part or assembly
- Avoid getting the components that you do not need

You can also store prune settings with the assembly to enable yourself and others to return to the same subset of parts (working layout):

- For faster redisplay, NX I-deas uses "level of detail" (LOD) technology. NX I-deas stores LOD with the model file. The first time you display a part, an LOD is generated. Thereafter, performance increases significantly
- "One-stop shopping" refers to the ability to build or change a structure, change attributes, prune an assembly, etc., all from one form: the hierarchy form. You can, for example, pick all of a particular part and modify all instances of it using only this form
- Constraints are displayed directly on the model for modification, deletion, information listing and status checking through the symbol's color coding
- Show Free gives an intuitive display of constraint status
- Users can easily create exploded configurations
- Animate dimension value changes to show a range of motion
- Users can "drag" dimension values for real-time motion studies
- Users can compare different versions of an assembly, before or after updating, to determine changes to hierarchy, constraints or part geometry
- Drawings created using NX I-deas Master Drafting are automatically updated when the assembly or the parts within it change

Prerequisite Master Modeler

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