



APPLIED CAX

NX large assembly tips and tricks

Mastering large assemblies in NX

Large assemblies in NX can involve thousands of components, intricate relationships, and complex dependencies, making them challenging to manage. Properly handling large assemblies is critical to productivity and efficiency. Inefficient techniques can lead to slow performance, errors, or even require duplicated efforts. This guide will discuss some of the built-in techniques, like using lightweight representations and simplifying part loading structures.

This is all to help ensure smoother workflows, faster load times, and reliable design integrity. Practicing proper techniques not only boosts productivity but also minimizes costly mistakes in engineering projects where precision and scalability are paramount.

Assembly load options and the benefits

NX provides a range of assembly load options, such as Fully Loaded, Partially Loaded, Lightweight, and Structure Only, to optimize performance and manage large assemblies effectively. These options allow users to balance detail and efficiency, from loading complete geometry for editing to using simplified representations or just the assembly hierarchy for faster navigation. Customizable settings further enhance flexibility, ensuring engineers can tailor the process to suit complex design demands while maintaining system responsiveness. Here are a few recommended load option settings:

Load structure only:

Loads the assembly hierarchy (component tree) without geometry, allowing quick review of structure and relationships without taxing system resources.

Partial loading:

Loads only a subset of component data (e.g., lightweight representations or reference sets) while keeping other parts unloaded until accessed. This balances performance and functionality for medium-to-large assemblies.

Lightweight representations:

Loads simplified, faceted representations of components instead of full geometry, significantly reducing memory usage and speeding up navigation in massive assemblies.

References sets

NX reference sets are a powerful feature in Siemens NX. They allow users to control which elements of a model (geometry, datums, sketches, etc.) are displayed or included when that part is used in an assembly. Essentially, they act as filters, letting you define specific subsets of a part's data, such as only the solid body, a simplified version, or specific features, to streamline assembly management and reduce complexity. Reference sets can improve performance by limiting loaded data, ensuring consistency across designs, and simplifying collaboration by showing only relevant information to downstream users.

Simplify components:

Define simplified reference sets for complex parts (e.g., removing internal details) to reduce the amount of data processed in the assembly.

Control data with reference sets:

Create and use reference sets (e.g., "MODEL," "SIMPLE," or custom sets) to filter the data loaded for each component. For example, a "MODEL" reference set might include only essential 3D geometry, excluding datums or sketches, to improve performance.

Arrangements

NX arrangements allow users to define multiple configurations or positions of components within an assembly. This enables the same assembly to represent different states, like open/closed or alternative layouts. In large assemblies, this feature is invaluable as it reduces file clutter, simplifies management, and ensures all configurations are tied to a single master model, maintaining data consistency.

Configure positions:

Use Assembly Arrangements to define different configurations or working positions of the assembly. This allows you to switch between views or states without loading unnecessary data.

Constraint management:

Set constraints as arrangement-specific to activate or deactivate them as needed, minimizing computation for unused configurations.

Simplify geometry

NX tools like Linked Exterior, Wrap Assembly, and Simplify Assembly simplify large assemblies by creating lightweight representations, such as external face copies, wrapped envelopes, or unified solids. They help reduce detail and improve performance. These tools hide irrelevant detail reduce the number of features to load, and accelerate processing and increase efficiency.

Assembly envelopes:

Use tools like "Wrap Assembly" or "Linked Exterior" to create simplified envelopes that represent sub-assemblies, hiding unnecessary details while improving load times.

Simplify assembly:

At the assembly level, apply the "Simplify Assembly" command to reduce geometric complexity (e.g., removing small features like fillets or holes) for parts that don't require full detail in the assembly context.

Proper modeling practices

Clean modeling practices in large assemblies ensure that parts are well-structured, free of errors, and optimized—preventing issues like broken constraints, excessive file size, or slow performance that cripple complex designs and assemblies. This discipline saves time, reduces troubleshooting, and maintains reliability as assemblies scale, making downstream processes smoother and more efficient.

Clean models:

Ensure individual parts are modeled efficiently (e.g., avoid overly complex sketches or redundant features) to reduce the overall assembly overhead.

Suppress unnecessary features:

Suppress features or components not required for the current task to lighten the session.

Summary

In mastering large assemblies in NX, leveraging tools like assembly load options, reference sets, arrangements, and geometry simplification, alongside disciplined modeling practices, transforms complexity into opportunity, ensuring efficient workflows, robust performance, and precise outcomes.

By adopting these techniques, engineers can confidently tackle intricate designs, minimize errors, and drive productivity in even the most demanding projects.