10 Best NX CAD Practices, Tips & Tricks

by Matt Martin, Mechanical Design Engineer at Sherpa Design and Applied CAx

In this article we'll discuss best practices in CAD, and NX CAD in particular. That said, best practices have a tendency to evolve and change over time (or at least they should).

Good CAD software will have new updates, changes, and tools in every release. Work flows and methods that once made sense long ago may no longer apply and new tools may require re-evaluation of current practices.

Let's go ahead and look at 10 best current tips and tricks...

1. Start with a Plan

For an angle bracket to hold a couple of components relative to one another, you probably don't need much of a model plan.

Sketch > Extrude > punch a few holes > make sure it fits > DONE…

But for a central part where many components attach to it, the creation of a model plan prior to starting in CAD can be very helpful. Decide in advance any necessary sketches, expressions, part functions, installation requirements, future part servicing, etc.

As well predict how the part may change over time and prepare for that change by building it into your model plan and ultimately, your CAD model. You can't anticipate everything and experience is key. Look at prior, similar designs and any issues collected from manufacturing and service. In two years when a revision is needed you, your colleagues and your clients will thank you for it.

Though they do have their place, generally speaking primitives like Blocks, Cones, Cylinders and Spheres should be avoided. Which is a good transition into…

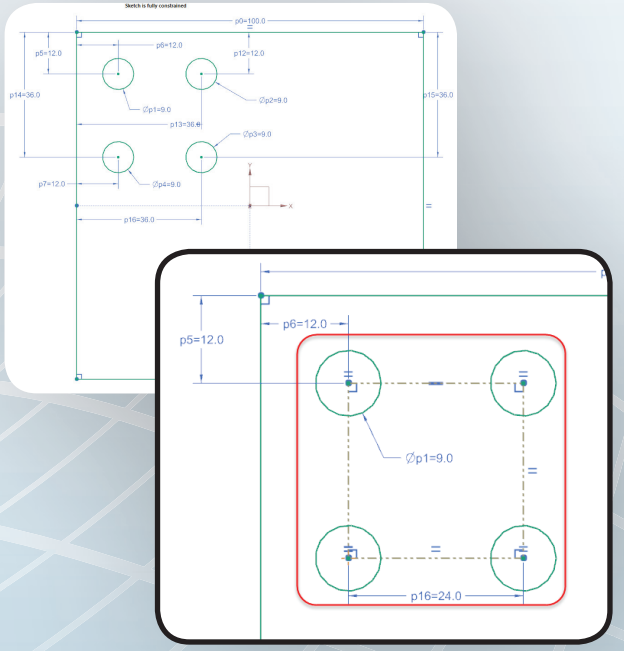
2.

Begin with a Sketch

Normal NX CAD workflow starts with a sketch section, open or closed, that is Extruded or Revolved for prismatic shapes. Sweeping, lofting & sweep along guide curves are also available for more organic or freeform shapes.

Here's some quick tips to keep top-of-mind as you're sketching:

- Sketches should be fully constrained with notable exceptions for motion or 2D kinematic studies.
- Choose geometric or relational constraints over dimensional constraints for faster updates.
- See the patterns:
    - If you're mounting a component that has a bolt pattern, create the bolt pattern as a grouping instead of individual holes with separate dimensions. Updates will be much easier and you won't end up with one bolt hole 1/4 diameter off and not caught until on the floor being assembled.
    [Figure at right]

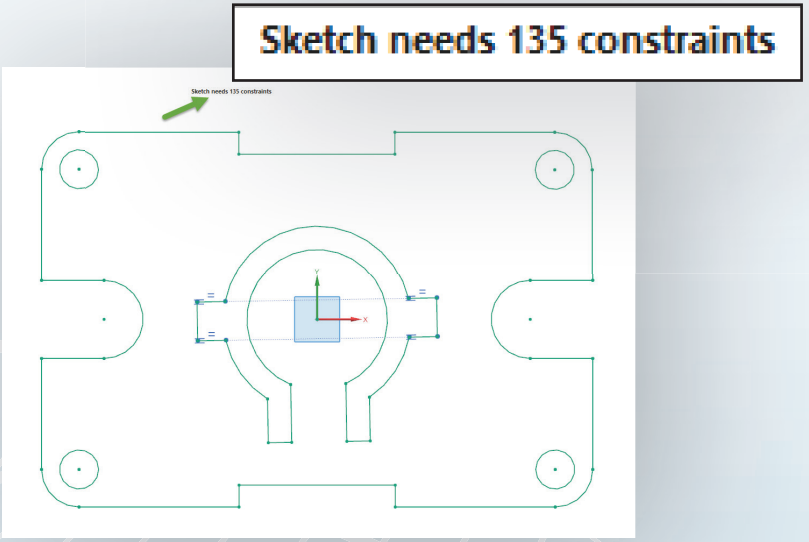


2.

o By using equal radius, equal length and construction lines, you can reduce the number of dimensions placed on a sketch making it easier to work with and convey your design intent.

- Minimize fillets or rounds in Sketches. Though fillets in sketches are more robust than the early days of NX, fillets, rounds and chamfers are typically best left to near the end of the design cycle.

- If your goal is to create a complicated, fully constrained sketch, I recommend you sketch a little, constrain a little, repeat. It is not uncommon to get the design you want only to see that you have hundreds of constraints to apply before you reach your goal. [Figure at right]



- For sketches later in the design, avoid referencing detail features like the tangent edge of a blend or sharp edge of a chamfer with sketch positioning dimensions.

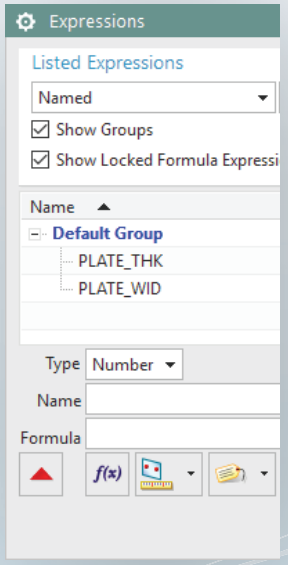
- 2D is not dead. NX Layout allows you to quickly capture and iterate your designs in 2D with fully parametric sketches.

3.

Expressions are your Friends

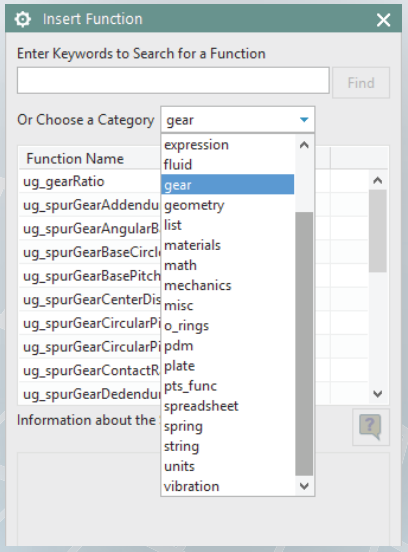
Use Expressions to create common arithmetic or conditional formulas to define your NX part. Expression lists are being generated in the background for every typed-in value in NX, so basically you're using them and you might not even know it!

- Every typed-in value creates an unnamed expression. An example would be a p223 = 25.6. Then that p expression could be turned into a named expression.



- Control the overall Height, Length, Width, Wall_Thickness, Diameter, etc. by using named expressions in the expression list. These can then be called when you create geometry. [Figure at left]

- Use **Extended Text Entry** and **Insert Function** and create complex relationships and formulae. [Figure at right]



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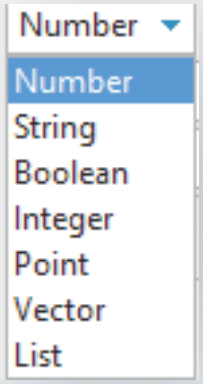
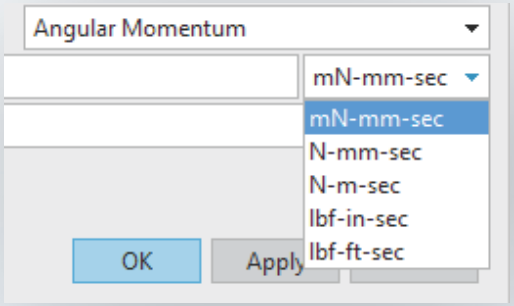


- Expressions can also allow you to create different types of expressions with appropriate units (area, volume, mass, etc.) [Figure at left]

- The Length type expression, for instance, can have units from km to angstrom and nearly everything in between.
[Figure at right]



- The NX expression system is flexible allowing you to input not just numbers but Strings, Booleans, Integers, Point, Vector and List. You can connect part attributes to expressions to drive downstream rules based engineering as well as downstream applications.
- In NX, you're using them regardless; use them to your advantage.
- Where do you find it? **Tools > Expressions** or **Ctrl+E.**

4. Extrude/Revolve with Intent

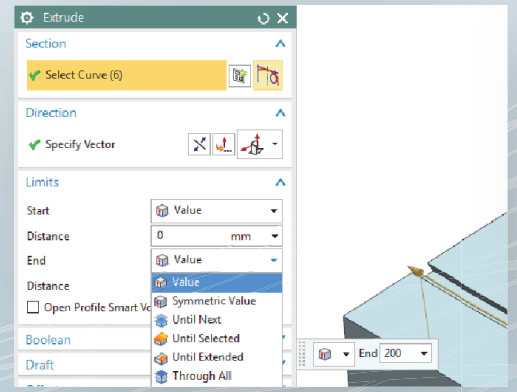
Leverage the tools available to you while creating geometry on the fly. This keeps your design intent no matter what the future changes of your design. Example A touches upon geometric constraints, Example B touches upon a poorly conveyed design intent, and Example C would use dimensional constraints driven by expressions (see the previous tip #3 regarding expressions).

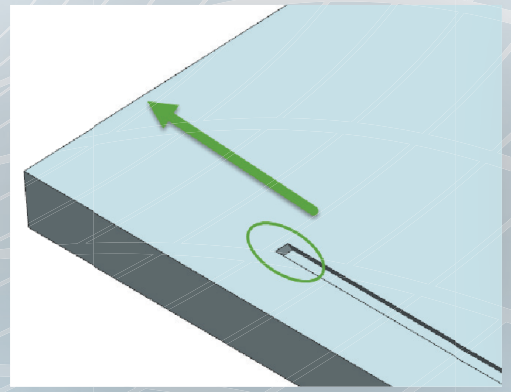
Let's look at our examples:

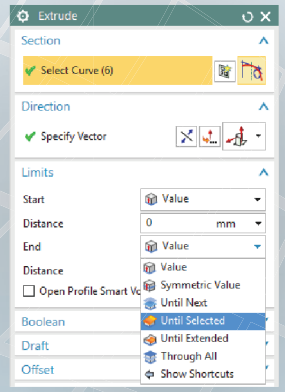
A. An example of properly conveyed design intent: a plate with a slot, by using **Until Extended, Until Selected** & **Through All**, you can convey design intent by stating that a slot the length of the plate will pierce the starting and end faces, regardless of the length of the plate.

B. But to use an example of poorly conveying design intent: If the plate is 600mm long and you create a slot feature that is also 600mm long, it is possible that the plate, when updated to 800mm long, would leave the slot at 600mm, no longer piercing the far-face.

C. Alternatively, you could use a PLATE_LENGTH expression to control both the length of the plate and the length of the slot. See the previous tip #3 regarding expressions.





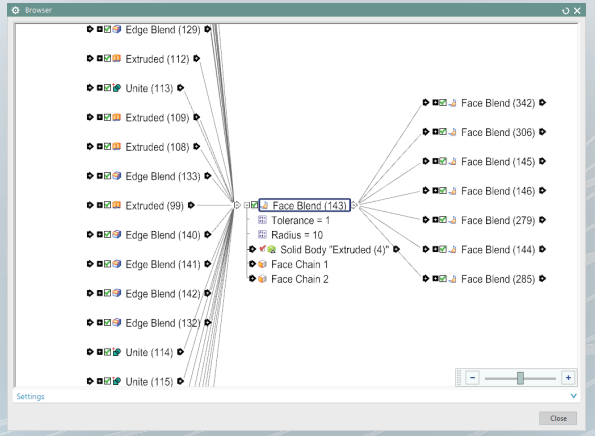


5. Modeling

There are two core approaches for Modeling. One is creating your geometry at 0,0,0 for all components that will go into parent assemblies. The other is to be in X-coordinate space where X is the parent controlling coordinate. For instance, aircraft- or vehicle-coordinate space. Let's begin with where to start your part!

- The primary or first feature should be related to the Datum Coordinate system at 0,0,0. This is typically the case for any part that can be used in multiple locations or in different assemblies, regardless of the coordinate space you're working in.

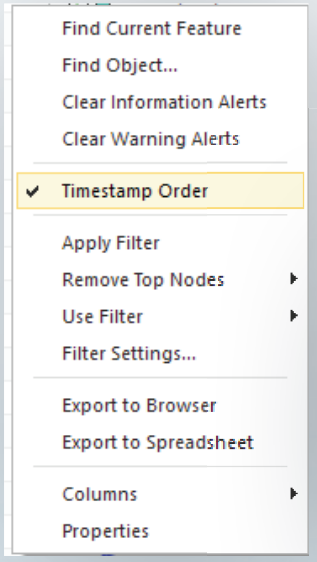
- If, however, you're working on a unique part, in coordinate space, the primary feature will likely be a Datum Coordinate system at some other location and orientation based on its location in the parent assembly. This is typically the case with aircraft manufacturers as well as automotive, shipbuilding and any other large structures. This way, you don't have to manage top level assembly constraints as the part is built "in-position."



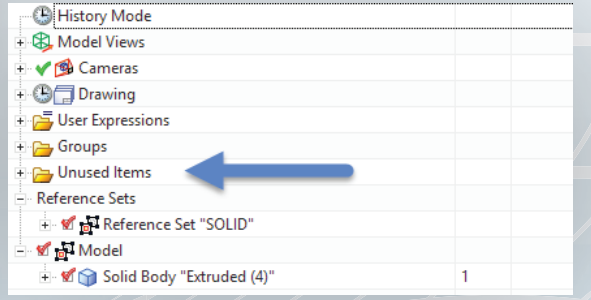
- NX is a history-based modeling tool where the order of creation is important. Generally speaking, the overall shape should be defined before placing in detail features such as threaded and counter-bored holes, blends (fillets & rounds), chamfers, etc.
[Figure at left]

5.

- Avoid buried features. A buried feature is one that is completely removed by another downstream feature. A set of tapped holes removed by an extrude/subtract clearance hole. An exception to this rule may be modifying an existing part for production. You may wish to show the part as it exists today with the necessary modifications ‘on top’ of them to convey intent. However, the released revision of this part should have the buried features removed. It just adds overhead to the creation of the part.



- Create only the geometry you need in the part file. Eliminate unused items by turning off Timestamp Order and examining the Unused Items folder. Again unneeded geometry is just extra overhead you don’t need (or want).
[Figures at left and below]



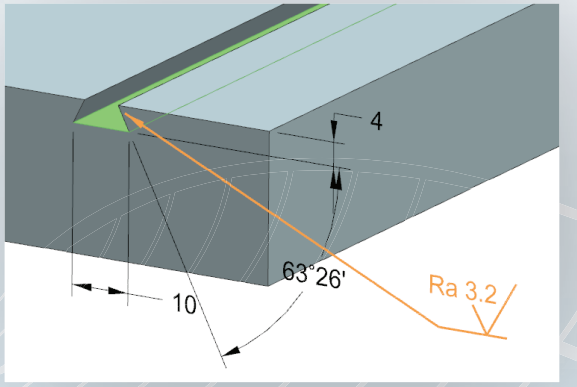
- If you have a choice of using a Datum or existing geometry, pick a Datum. If you’re not picking a Datum, you should reference geometry that is less likely to change or get deleted over time.

6. Design for Downstream

Are you vertically integrated? Do you manufacture, assemble and ship the parts you are designing within your company? Do you have a tightly integrated supply chain?

If so, designing for downstream utilization is becoming more and more important, not only for internal consumers of your data but external supply chain. You want to be thinking about designing for excellence, or to be blunt, Design for X where X is:

- Design for Manufacturing
- Design for Assembly
- Design for Service
- Design for Revision
- Design for Recycling



No longer can an engineer design a part to “throw it over the wall and let them figure it out.” Thinking about the whole ecosystem is going to make your part cost-efficient and more reliable. Let’s cover some of the details involved with this line of thought:

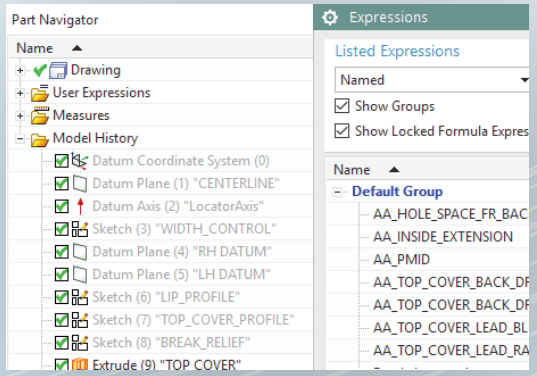
6.

- As engineers and designers working in CAD we understand what a virtual 3D part is. Designing for downstream starts with the **CAD model** and the model plan. Typically, parts are designed to a net shape without regard for ± tolerances, materials and finish. In the past, part material, tolerances and finish were applied to the drawing. This information sat in a text note on a PDF or TIFF drawing in some database and is not accessible to anyone unless they open the file and read it. More often than not, this information was available to the design engineer at the time of model creation. It makes sense to place that information on the model at the time that you know it for downstream uses.

- Applying part attributes for Material, Finish and utilizing Model Based Definition or MBD can convey a wealth of information to downstream users.

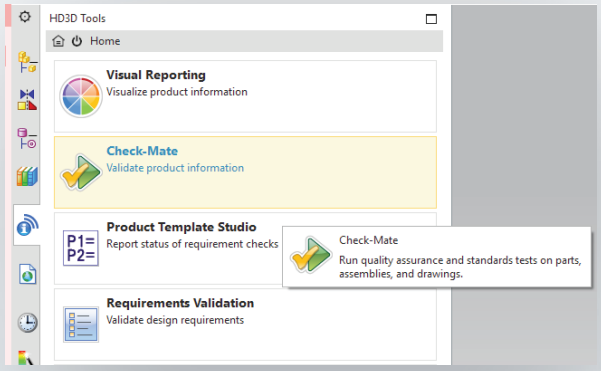
- In NX CAD, application of MBD is known as Product Manufacturing Information or PMI. PMI can reduce the amount of time to program a part for CNC fabrication.

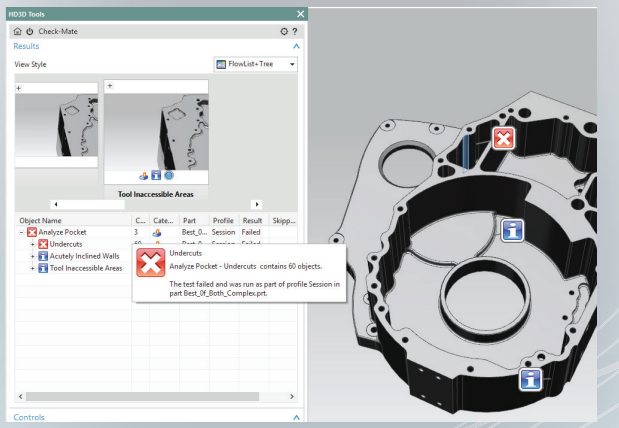
- Along the same lines, inspection programming time for Computerized Measuring Machines or CMM is greatly reduced with the application of MBD/PMI.



- Think about the person who's going to take over your model. Help a guy out! Name your features and expressions. Organize features in a logical and progressive order. If a feature is added late in the design cycle but is best placed early in the part history, use **Make Current Feature** to roll back the model, add the necessary changes, then **Update to End** and ensure all features update. Organize your features into logical groups with **Feature Groups**. [Figure at left]

6.





- Use available tools to ensure manufacturability, enforcement of design standards, alert designers to CAD model geometry issues and more with tools like **Checkmate**. Design and Requirements Validation

http://www.plm.automation.siemens.com/en_us/products/nx/for-design/visual-analytics/design-requirements-validation.shtml

- If the part you're designing must be installed with a particular orientation, use the concept of poka-yoke or mistake-proofing. This is particularly effective when similar parts are assembled together that, without poka-yoke, could be out of order. An example is clocking one bolt hole in a bolt circle a few degrees off of the typical interval, where if installed upside down the hole is blocked and there is only one way to install the part to the assembly.

https://en.wikipedia.org/wiki/Poka-yoke

- Patterning features will allow the downstream assembly to utilize **Pattern Component** to quickly populate fasteners and other components that need to reference the original part pattern.

7. Emphasize Reuse - Save Money

How many times have you designed and released a part only to find that it is nearly identical to an existing, previously released part? Or designed, from scratch, a complex part that 'could have been' a save-as from a similar part and modified to save time.

In today's hair-on-fire, gotta get that product shipped by end of ____ (fill in blank) leaves many engineers with little time to find an existing solution. In the best of times, designers will search for an existing part for 5 to maybe 15 minutes before deciding to introduce a new part number to the system.

The Aberdeen Group estimates that it costs between $4,500 and $23,000 per item for the introduction of a new part number. Duplication of effort to design a part typically pales, by comparison, to the cost of introducing that part to the supply chain as it may only take you 30-45 minutes to design, draft and ready a new part for release.

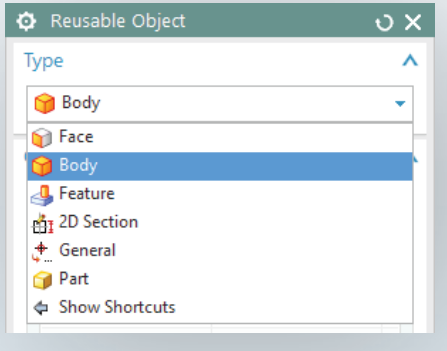
Let's look at some tips for enabling reuse:

- The use of a name rule system can help reduce the number of duplicates generated by your design team. Logical and consistent names presented to the user when they are searching for components will help. One of the biggest areas of duplication I see when I visit customers are simple fasteners. While there are many reasons for this (couldn't find the fastener needed, employer purchased another company and their parts were dumped into the system, etc.), it does require a level of discipline:
    - Names like SCREW,M5X25,SHC,SST is pretty easy to find as long as other screws are named similarly. Keep part names consistent.

7.

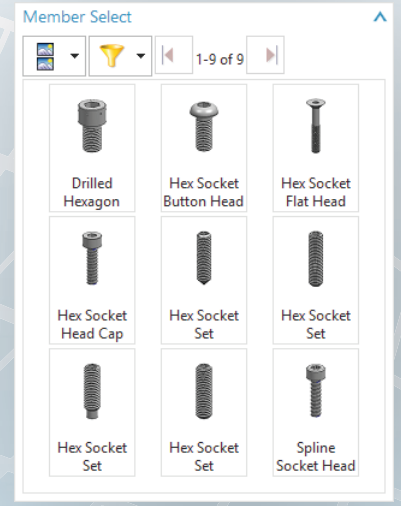
- Avoid abbreviations when possible.
- Develop an approved list of abbreviations, only if needed due to limits in data management, operating or CAD systems.
- Keep naming rules to help classify the part and go from generic to specific such as “BRACKET, LEFT, HIGH-RESOLUTION OPTICS, MR5530”

- Analytical tools are available to help reduce geometric duplicates like Siemens PLM Geolus Search. Another approach is Classification within Teamcenter.
http://www.plm.automation.siemens.com/en_us/products/open/geolus/



- Reuse is not limited to the components in your database or file system. In NX you can reuse Faces, Bodies, Features, 2D sections and more. These will be organized by the Reuse Library. [Figure at left]

- NX Part Families should be used to generate all of the fasteners required for your organization. Creation of fasteners from a common ‘seed’ will facilitate easy component replacement and maintain relationships in the assembly. Part Families can be used for any table driven type of part as well as mirrored left/right parts. [Figure at right]



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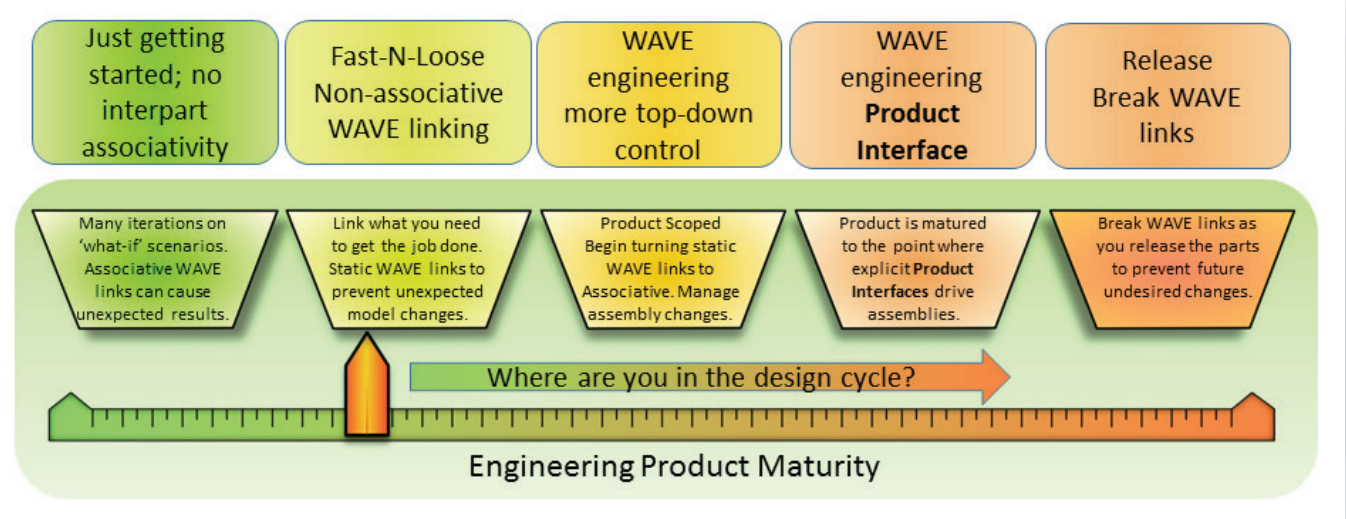


Associativity is not the only goal

Interdependent feature updates may give you surprise. Maintaining features independent of external changes during the initial engineering effort will speed you along in your design cycle. Interpart links like WAVE and interpart expressions can lead to unexpected results during initial project phases when a lot of "what if" scenarios are being presented.

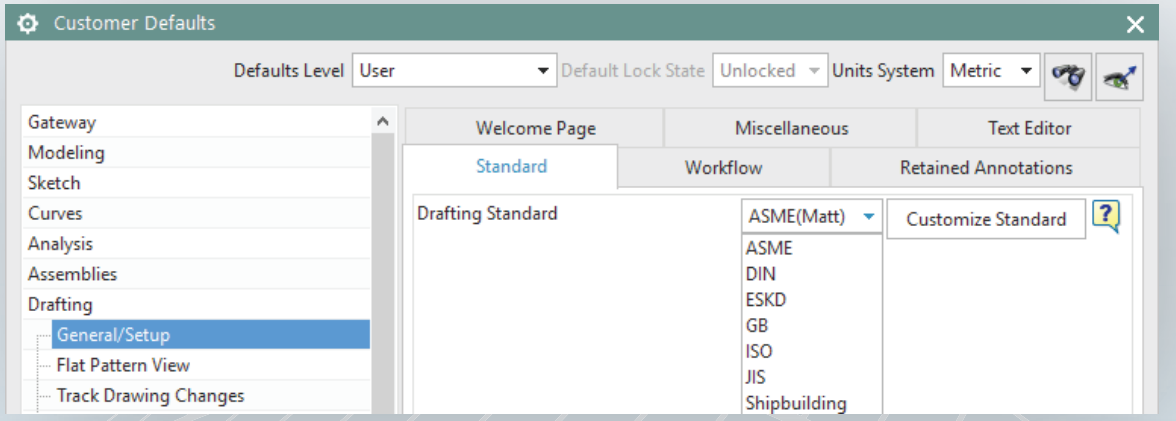
Early on in the design cycle, a 'fast and loose' approach will allow you to iterate quickly to eliminate designs that won't work. Non-associative WAVE linking, Synchronous Technology and push/pull type modeling allow you to iterate quickly and efficiently to determine the scope of the overall project.

Later in the design cycle is when you want to put in more and more constraints, and you look at associative WAVE linking, Product Interfaces, and top-down engineering.

9.

Drafting and MBD Standards

Communicating your designs to fabricate parts and build assemblies is, of course, the end result of design and engineering efforts. Clearly communicating the design to a fabricator or contract assembly vendor is key to getting the product you want to represent your company. Clearly defined standards make it much easier for your supply chain to read and interpret your 2D drawings or MBD/PMI.



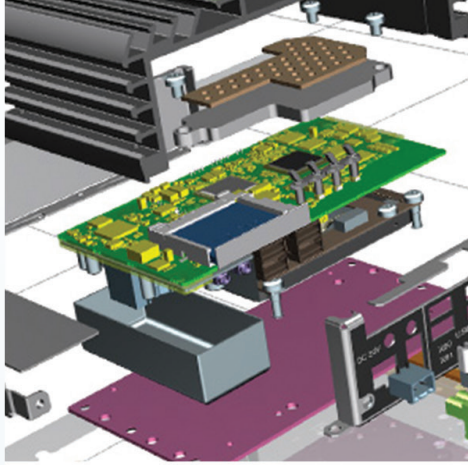
As the world is moving from a 2D, paper drawing approach to a 3D, Model Based Definition approach, the way we document a part is changing. It is much easier to interpret design intent of a 3D part with associated dimensions, callouts, notes, etc. As you pick a dimension or callout, the referenced faces, edges and features will highlight. A complex part may require several sheets of detailed drawing views, section and auxiliary views. That same part can be fully documented, in 3D, in just two or three MBD/PMI views. Less time documenting your designs means more time to iterate, explore and perfect your product design.

10.

Education is key

It is a time consuming task to find ways to stay up-to-date with changes in NX. Demands to get new designs completed and shipped directly affect your bottom line. Your team should develop internal Subject Matter Experts (SMEs) to pour through the What's-New documents in each release of the software, evaluate the impact on current work flows and suggest adjustments as needed. SMEs have intimate knowledge of how the tools are being used within your organization and can quickly find the new & updated tools that can help the daily design engineer. SMEs can head off acceptance issues when major changes are coming in new releases, train incoming personnel on standards and practices and continue internal training for the existing designer base.

Training can be especially important if your company skips a release or two before upgrading. It is not uncommon to jump two, three or even four released versions. In these cases, a What's-New in NX summary can be particularly useful.



Summary:

It's all too easy to get lost in the minutia of CAD design and documentation standards. Pick a standard like the ASME Inch/Metric, ISO, DIN, JIS, etc. and set the foundation for your design efforts moving forward. Establish an overall methodology and specific best practices within your organization and adhere to them. Develop SMEs to regularly review best practices and ensure you are using the best tools and most efficient work flows.

This is especially important if your company is new to NX CAD/CAM/CAE as there is a learning and maturity curve. As your team matures in the tool, they will look to extend their design and analysis capabilities to design better, more reliable products, faster. Hold regular user group meetings to share knowledge and best practices across your organization. Look to external resources, like AppliedCAx, to evaluate current workflows and CAD practices and get the most out of the tools you currently have in place.