

Why Synchronous Technology Matters

There is a considerable amount of discussion surrounding Siemens Synchronous Technology. The core questions appear to be: “What is it exactly?”, “How does it work?”, and “Why does it matter?” Ideally, this brief will answer those questions with clarity.

There are two implementations of Synchronous Technology in the market place. They are the Siemens CAD products, NX and Solid Edge. This brief will focus on the NX 6 implementation of the technology. NX 6 brings both a history and history-free mode of operation to the software, not seen in earlier revisions. Direct modeling, though present in prior NX versions, has been significantly enhanced and combined with Synchronous Technology to form “Design Freedom”. Design Freedom is the sum of traditional NX CAD capabilities, enhanced Direct Modeling and Synchronous Technology.

When a model is brought into NX, be it an imported solid body, surfaces, or a native model, the user has the option of performing modeling operations in either a history based or non-history based fashion. Regardless of which mode the user chooses, many features, such as blends, holes, shells and ribs, remain feature-based and can be modified as such. When NX is operating in history mode, ALL direct modeling operations are parametric, in that the parameters specified during the creation of them, remain available for subsequent modification, and/or for use in equations, just like any other traditional history based modeling operation. When in history mode, direct modeling operations are added to the history tree just like any other traditional history based, parametric modeling operation.

These attributes serve the dual purpose of presenting to the user both a history of what was done to the model over time, and the ability to decide whether it makes more sense to parse that history to perform a modification, add to it, or disregard it altogether.

If the user chooses to abandon the model history, doing so is simply a switch to non-history mode, which can be done in reverse as well, at any time the user chooses. Models may be switched between history and non-history mode at any time.

A model might start out as a traditional history based one and see significant modifications with synchronous, direct modeling operations. Then the design path changes in such a way that a rebuild of the model would be warranted if the only tools available were the traditional history based tools. At that point, the history can be disregarded in favor of a history free, feature based path. Some time down the road, the design path solidifies, once again warranting a change to the history based path, where parameters and automation may be applied to streamline further changes.

At all times during this process, an NX user enjoys the benefit of synchronous technology in that it parses the part geometry in real time, assisting with design intent, based on the part topology, selection

intent, and direct modeling operation in play. This design intent assist comes in the form of the software parsing the model topology and presenting smart choices, based on the implied design intent it sees. This process is not unlike the one we do when considering a model in our own mind!

This design intent assist also works with selection intent functions that make the task of selecting topology entities to manipulate more productive. Where direct modeling operations do free the user from having to keep model history features in consideration, it can be tedious to correctly and efficiently select entities. Direct modeling plus Synchronous Technology largely eliminates this trade-off.

This is unique in the industry right now, and something other CAD product vendors appear very willing to attempt to marginalize!

When a NX user is operating in the new non-history based mode, a model boils down to solid or sheet bodies, features such as holes and ribs, and dimensions that are solved variationally*, when the user modifies them. Direct modeling operations perform exactly the same way they do in history mode, and depend on selection intent and synchronous technology to assist in the specification of the operation itself, and in the context of the implied design intent, present in the model topology at the time. The user has control over the synchronous technology in that specific cases can be ignored, or favored over others where it makes sense to do so. The only real difference is that a model history is not created.

Should the NX user feel the need to use the traditional, sketch-based operations, they can do so in history-free mode as well. What happens in this case is the associative links, normally necessary to realize a coherent history tree, are simply not generated. The user then is free to build the sketch, make solids with it, move the sketch and or the solids, perform booleans, etc... without having to worry about managing the chain of references normally associated with those kinds of activities! When the model requires very aggressive changes, this is a huge time savings and, depending on the users overall level of skill, can be a good mental savings as well. Not having to manage chains of references makes operations like “cut and paste” features not only practical, but preferred!

**Variationally means features and constraints can be solved simultaneously, out of order. Traditional parametric, history based design is sequential, meaning the order of things matters, must be managed, and kept under consideration during the entire modeling process. Synchronous Technology is about being able to apply variational AND sequential techniques to the engineering problems at hand, no matter what system authored the model originally.*

With these combined technologies working together, selecting a feature from one model, performing the “copy”, or “cut” operation, then locating it and combining it with another model become simple and straightforward to do. The user enjoys a higher degree of model and feature re-purposing as a result.

All the traditional sketch based modeling operations perform in essentially the same manner in history free mode just like the direct modeling operations do, whether or not the user is operating with model history. This means the choice is about realizing the most effective modeling path at all times.

Given these things, it is just as possible and practical for a NX user to start with a native authored model as it is with non-native data. No matter where the data came from, or what tool it was authored with, the full range of design path options are available, leaving them free to choose the path that makes the best sense. This means less time spent planning and sorting out which modeling tool operations make the best sense -vs- having to make compromises in their engineering work flow. That is a very significant value add these days, given the variety of CAD systems in play, and the growing need to incorporate model data designed in various systems into the engineering workflow.

All of this functionality is built into the core of NX, thus extending it to related activities, such as manufacturing, analysis, detailing, rule based knowledge engineering, and other tasks. Synchronous Technology isn't just about modeling operations; it is all about enabling the best engineering process workflows.

One other point to make clear is the difference between history based parametric and variational parametric technology. With the acquisition of SDRC, Siemens inherited both best in class variational design technology and data management technology. This can be seen in the NX product as part of Synchronous Technology and Design Freedom, and the much improved Teamcenter, cross platform, multi-CAD capable Teamcenter and Teamcenter Express products.

The multi-CAD capabilities of Teamcenter enable the management of complex data from multiple vendors and the Synchronous Technology present in both NX and Solid Edge mean being able to make effective use of that data at a level on par with native models, no matter which CAD tool authored the model. Those capabilities are unique in the industry.

Parametric variational modeling has what can be described as feature groups, instead of a feature tree. The features are solved simultaneously, not in sequence. When NX is operating in history free mode, features, such as holes, ribs, shell and other operations then remain feature based with easy to modify parameters, while not having the burden of chains of history references associated with them.

On an assembly level this difference means being able to enjoy considerable freedom in how, or if various levels of an assembly are to be constrained fully, and constraint networks can be localized to those areas of the design that actually benefit from parametric changes.

Large scale assembly design without these capabilities is a tedious and time consuming exercise that also demands a stiff computing penalty as well. The human cost savings, in terms of realizing all the degrees of freedom, is substantial when compared to ordinary parametric assembly functionality.

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Rather than take an “either/or”, or “bolt on” approach, Siemens has acquired the core enabling technology elements necessary to build a holistic environment where the users have the full array of choices available to them at any time. This remains true in NX whether the model is foreign or native, or if the user is in forward create design mode, or modify mode.

The significance of this is being able to leverage the best modeling strategy applicable to the problem and working relationships in conjunction with all other CAD models. History based, parametric modeling still enjoys a lot of value for many classes of modeling problems. Two very easily seen examples are Part families and automated designs. Direct modeling and feature based modeling present as very potent solutions during the concept or modification stages of modeling, but lack sufficient automation capability to be leveraged as the product being designed matures, or spawns derivative designs.

There is nothing about Design Freedom and Synchronous technology that devalues traditional history based parametric modeling. Instead, value is added by enabling the users to make the choices they need to make, when and how they need to make them, instead of having to choose from a limited set of choices made available through their software tool.

All of this is built into the foundation of NX 6. No new license is required, nor does it demand that the user abandon familiar and effective means and methods. What it does mean is a significantly greater ability to work with any CAD data, and realize design needs quickly and efficiently.

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