CAE geometry cleanup and multi-CAD support

Rapid analysis model preparation using synchronous technology for faster design-analysis iterations

fact sheet

Siemens PLM Software

▶ **Summary**

NX™ software's CAE solutions include powerful NX capabilities, such as multi-CAD support and synchronous technology. By using NX, you can rapidly clean and prepare geometry from any CAD source for your analysis modeling. Geometry edits and the complete analysis model remain associated to the base design, which means you can easily update your analysis model each time the design changes. As a result, NX accelerates your design-analysis iterations and improves your overall productivity.

Benefits

Save time by speeding up geometry cleanup and analysis model preparation

Improve quality by facilitating faster design-analysis iterations, which enables you to simulate more design alternatives and deliver better products faster

Lower software cost with a single CAE environment that integrates all of your design solutions

Lower training cost with intuitive interface that enables analysts to easily learn geometry manipulation and analysis modeling

Features

Synchronous technology for fast, intuitive geometry cleanup and analysis model preparation

Geometry edits and analysis model associative to base design, enabling prior geometry edits and analysis model to automatically update as the base design changes

Multi-CAD support from a variety of sources, including the ability to edit all multi-CAD geometry using synchronous technology

Support for multiple geometry data formats, including: JT™, IGES, STEP, I-deas® software, Catia V4, Catia V5, SolidWorks, Solid Edge® suite and ProEngineer

NX synchronous technology has been widely recognized since its launch as a way to revolutionize CAD design. However, synchronous technology is not just limited to CAD. Just as importantly, it can save significant time across your CAE processes.

Geometry challenges with standalone CAE preprocessors

One of the main issues you face as an analyst is the need to clean and de-feature imported geometry. Features like small holes, small steps and surfaces impact meshing speed and quality but can be irrelevant to engineering performance. On average, *for each design iteration*, engineers spend an entire day or more on geometry cleanup activities when using standalone CAE preprocessors. This wastes time that could be better spent doing real engineering.

The source of the geometry data itself poses an additional challenge. CAE preprocessors usually do not support all CAD data equally. This results in data translation issues, lost or inaccurate geometry and ultimately in the need for even more manual geometry cleanup.

Applying synchronous technology to CAE

By using NX for pre-processing, you can leverage synchronous technology to clean and de-feature geometry in a fraction of the time it takes for traditional CAE tools to perform these tasks.

Figure 1 illustrates a situation where an analyst needs to remove a small surface from within a cylinder. This small surface is unimportant since it will result in very thin, poor quality elements. Since this design feature is not critical to analysis, it should be removed.
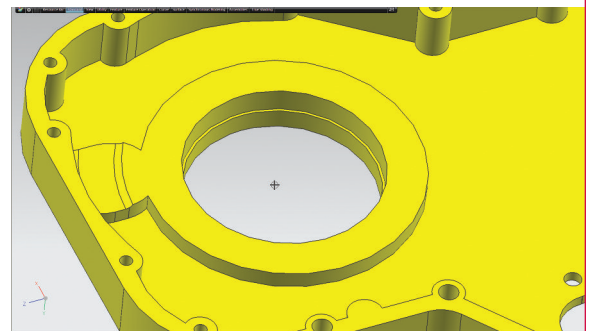

Figure 1: Cylinder with small surface that needs to be removed.

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With NX, you only need to perform three simple clicks to complete this task:

- Select the *co-planar* command from the synchronous modeling toolbar (figure 2).
- Select the surface that you want to move, in this case the small inner surface.
- Select the stationary face, which is the top surface of the main cylinder.
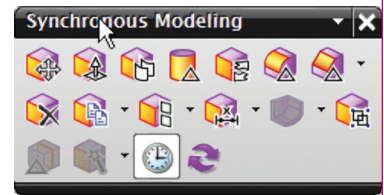- Click OK.

Figure 2: Synchronous modeling toolbar.

After clicking OK, the thin face is made co-planar with the top face, thereby removing the step within the middle of the cylinder (figure 3).

The cleanup procedure in this example only takes seconds in NX. Other CAE preprocessors could have taken hours to edit this geometry.
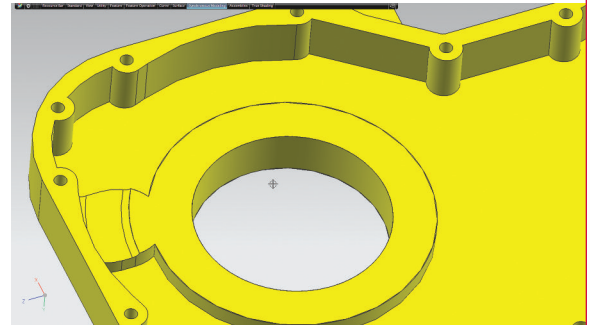
Multi-CAD support

Figure 3: Cylinder after small surface has been removed.

The source of the CAD data does not matter in trying to accomplish the task just discussed. Even if the designer revises the geometry outside of NX, you only need to import the latest CAD revision into NX and your prior synchronous technology geometry changes will automatically be applied to the new design revision. This saves you a significant amount of time, not just in a single design iteration, but in all subsequent iterations as well – thereby improving overall product development efficiency.

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