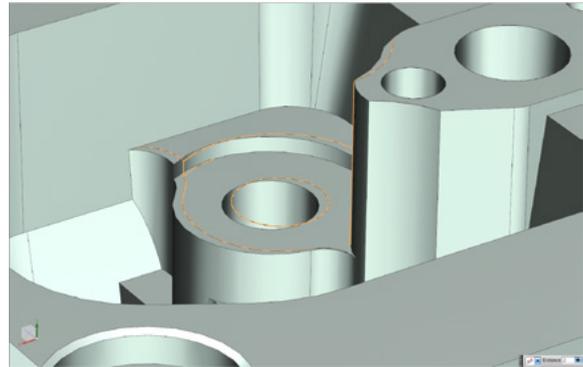


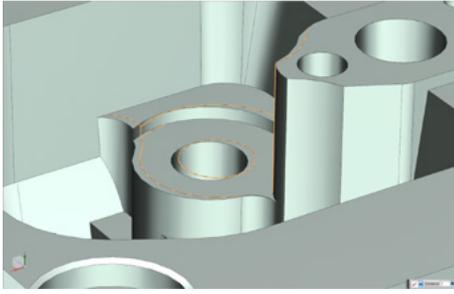
NX 9

WHITEPAPER

Using Synchronous Modeling to Manipulate Solid Bodies in NX



Using Synchronous Modeling to Manipulate Solid Bodies in NX



Software:
NX 9 Design

Overview

Synchronous modeling allows manipulation of solid bodies without a need for a history or feature tree. Solid geometry imported from a Step, IGES, Parasolid, or another CAD system can be manipulated using commands like Move Hole, Delete Feature, and Offset Face. Understanding the power and simplicity of synchronous modeling is paramount when designing in NX.

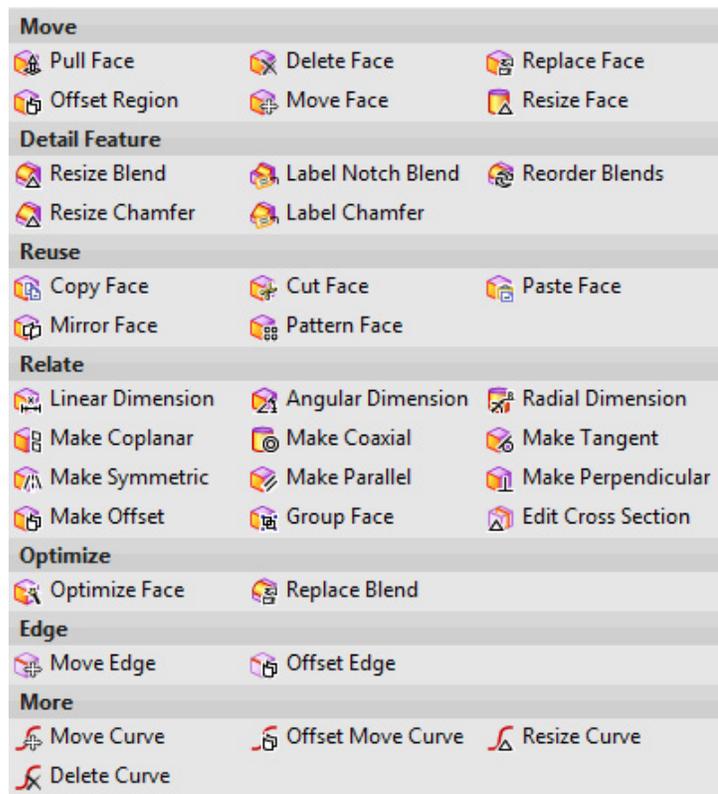
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Synchronous Modeling Tools

Synchronous modeling lets users make changes to models without knowing feature relationships or dependencies. Changes to the model are preserved for downstream processes like simulation or manufacturing and are tracked with a feature in the feature tree. The synchronous modeling tools have been expanded with new releases of NX, with edge and curve manipulation tools added in the NX9 release. The edge, face, feature, and curve manipulation tools create a powerful and efficient synchronous modeling suite. The synchronous modeling toolset is shown in Figure 1.

Figure 1: ►
Synchronous modeling toolset.

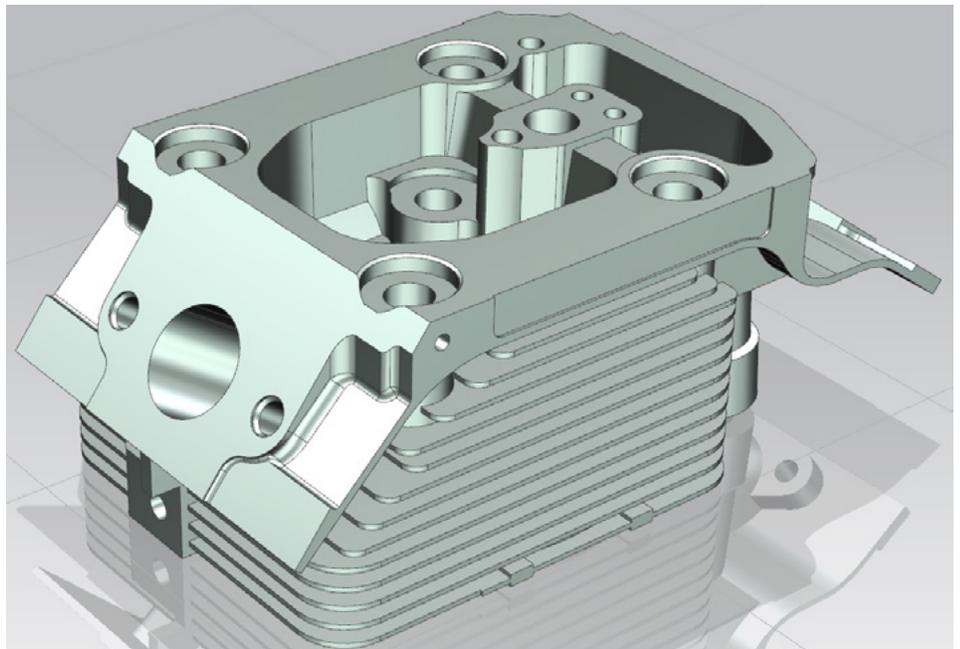


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Synchronous Modeling Functionality

Detailed CAD models often contain hundreds of features. Without knowledge of the parent-child relationships of each individual feature, engineering changes are difficult, and the original designer is often the only person who expertly knows the model. Alternatively, a vendor model is imported from a generic format such as Step or Parasolid and does not contain a feature tree and Editing this solid body with cuts and extrudes is tedious and may be impossible. In these cases, synchronous modeling tools make design changes simple and efficient. Whether using a model with a full feature tree or a single solid body, synchronous modeling tools integrate easily with the current design and downstream processes. A typical solid body received from a vendor is shown in Figure 2.

Figure 2: ►
Single solid body.



This model contains many complex features, making it difficult to make a design change without breaking the CAD model, but synchronous modeling makes these design changes simple. Let's try to move a mounting hole boss in the center of the part. It would be difficult to move this boss with basic cut and extrude features. Figure 3 shows how this engineering change can be made using the Move Face command under the Synchronous Modeling toolbar.

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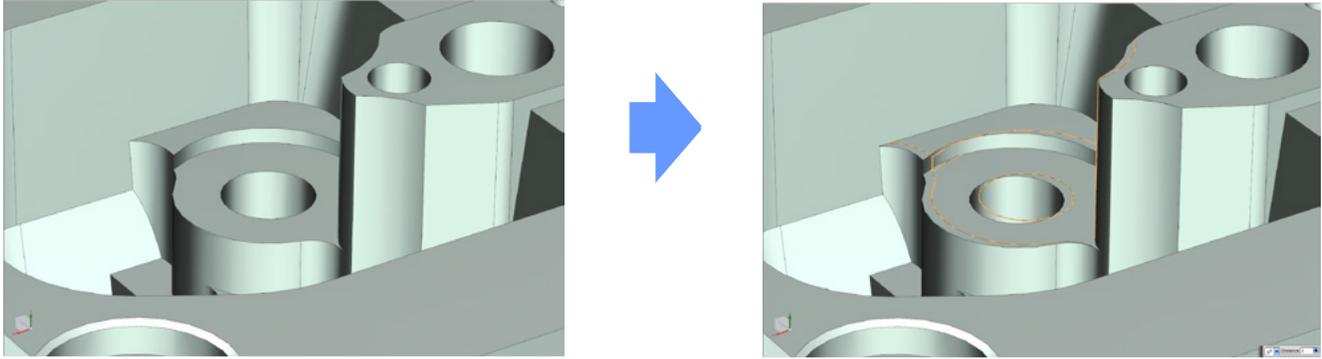


Figure 3: ▲
Moving boss with Move Face command.

The orange outlined faces from the original model were selected in the Move Face command. A preview shows the end result of the model after the command. All of the selected and related faces were updated to reflect the moved boss. A Move Face feature was created in the feature tree and can be edited later.

Using the same base model, let's try to change the angle between two faces. This operation would be difficult to perform as a simple cut because the beginning and end faces are not parallel and contain fillets. Figure 4 shows the change using the Radial Dimension command, which is also found in the Synchronous Modeling toolbar.

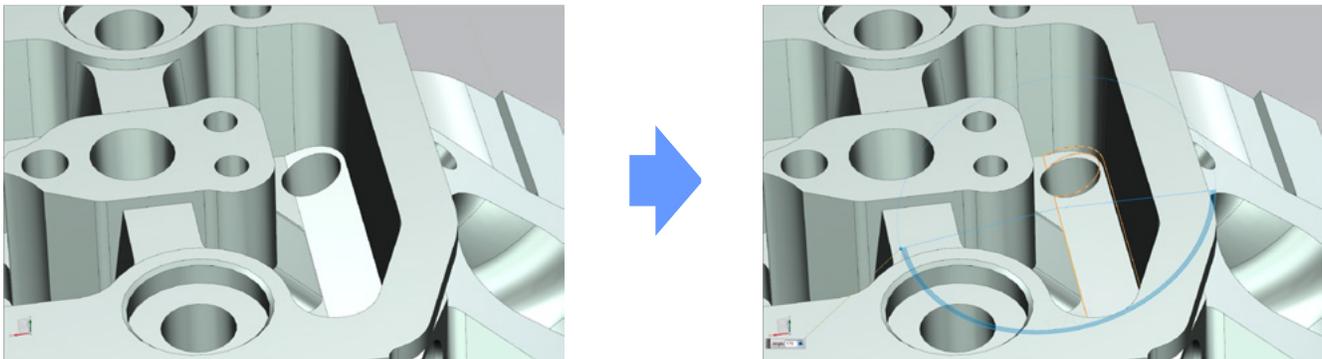


Figure 4: ▲
Adjusting angle between faces using
Angular Dimension command.

Again, the highlighted face was selected with the Angular Dimension command. The preview shows how the resulting model will look. This command works well even when the face is defined by multiple hole and fillet features. The angular dimension becomes a feature in the feature tree and can be edited later.

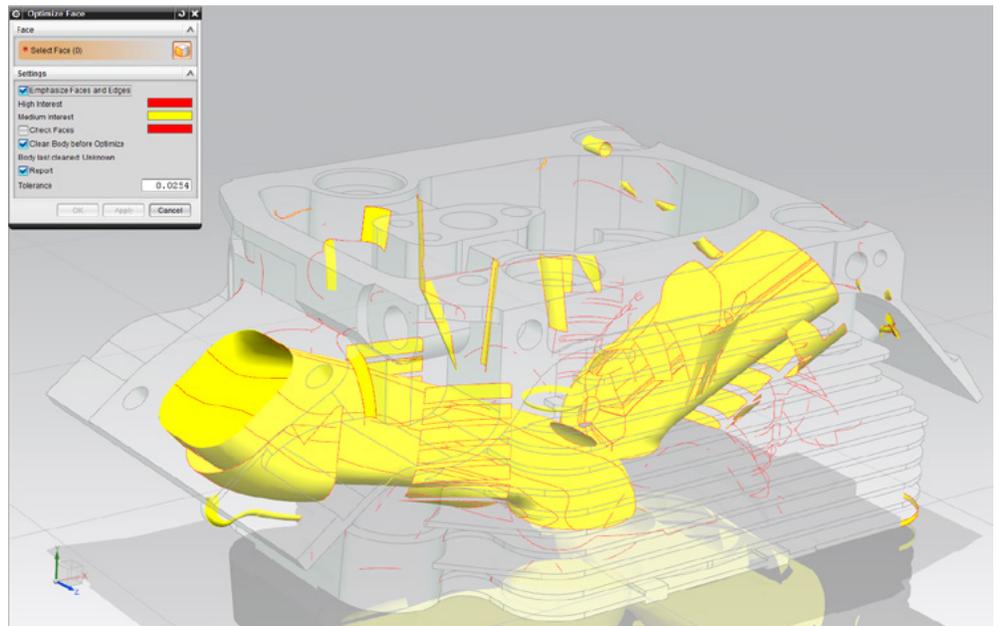
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Synchronous Modeling Best Practices

Synchronous modeling is very powerful and efficient when used properly. Here are some general tips to effectively use the synchronous modeling tools.

1. Vendor models often come into NX as solid bodies that lack feature trees; such models contain an array of faces and edges, all of which are characterized by NX. Sometimes, a blend from another CAD system may be interpreted as a b-surface by NX. It is good to understand how NX has defined the faces and edges in your model. The Optimize Face command from the Synchronous Modeling toolbar highlights faces NX considers to be of interest. When the Emphasize Faces and Edges checkbox is checked, NX highlights faces and edges that are not analytic (e.g., b-surfaces or splines) and may be difficult to perform synchronous modeling on. When the problem faces are selected and the Optimize Face command is applied, NX will attempt to simplify highlighted entities (Figure 5).

Figure 5: ►
Faces NX deems of interest.



Using Synchronous Modeling to Manipulate Solid Bodies in NX

A few notes on Optimize Face:

- The Optimize Face command re-parameterizes your model and therefore deletes all model history. The resulting feature will be a body.
 - Be careful of surfaces or curves you do not want simplified, such as fluid domains that contain specific b-surfaces or splines. NX may try to oversimplify them.
 - There is an Optimize checkbox on the geometry import command. The Import Optimize and Optimize Face commands perform similar tasks but sometimes produce different results. The Optimize Face command is more robust than Optimize on Import.
2. Depending on the type of file imported (e.g. Step, IGES, Parasolid, or another CAD system), the solid geometry may vary slightly. For example, a part imported as a Parasolid file may contain more sliver geometry than the same part imported as a Step file. In general, Step files produce the highest-quality geometry when importing into NX.
 3. Synchronous modeling contains many overlapping tools. For example, the Move Face and Pull Face commands produce very similar results in certain situations. If one tool is not working as desired, try another tool. Also, make sure to inspect the locations where the synchronous modeling tool is failing. There may be a small modeling error from import that is causing the failure. Finally, make synchronous modeling changes in small increments. This allows NX to heal a small number of issues at a time and will often allow larger overall changes.

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