

Unigraphics NX 6.0 Tutorial Update

- An Accompaniment to the Tutorial "NX5 for Engineering Design"

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How To Use This Document:

This document is meant as an accompaniment to the tutorial "NX5 for Engineering Design" by Mign C. Leu and Akul Joshi. It is not meant as a standalone reference.

Important and significant version changes between Unigraphics NX release 5.0 and 6.0 that may cause frustration to users attempting to complete the tutorial for NX6 are detailed in this document. Updates are not exhaustive in that some discrepancies are not noted for multiple tools in which discrepancies are identical. Updates noted below contain only information that is **not** contained in the original tutorial. Any still relevant tutorial material will not be reproduced in these updates.

Acknowledgements

We gratefully acknowledge the efforts that Professor Mign C. Leu and Mr. Akul Joshi have put into the original document "NX5 for Engineering Design" and their generosity to share their excellent tutorial with us.

UPDATES:

2.1.1

From the Windows desktop screen, click on Start --> programs -->UGS NX 6.0-->NX 6.0.

2.3.2

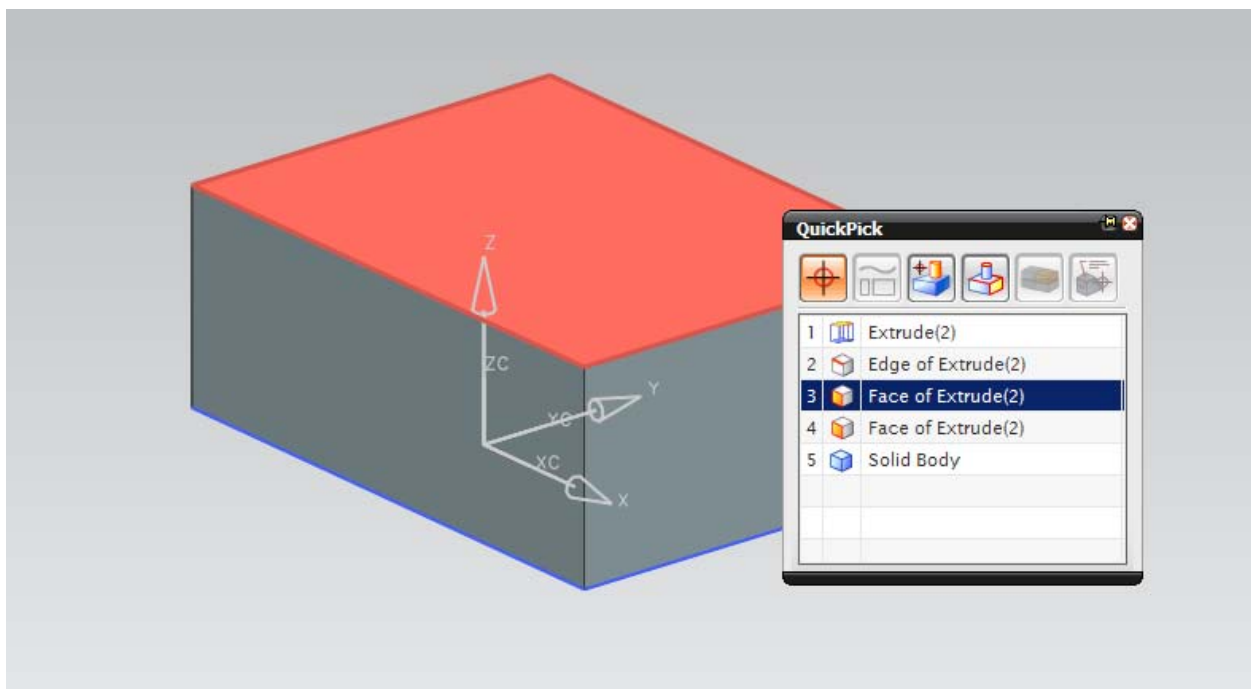
Image of Gateway incorrectly labelled - resource bar label points to manufacturing wizard. The resource bar is the entire collection of menus on the left hand side of NX. The entire resource bar may be undocked with the pin icon as shown in the tutorial. Undocking the resource bar causes it to hide until icons are rolled over:



2.3.3

GENERAL OBJECT SELECTION (pg 24)

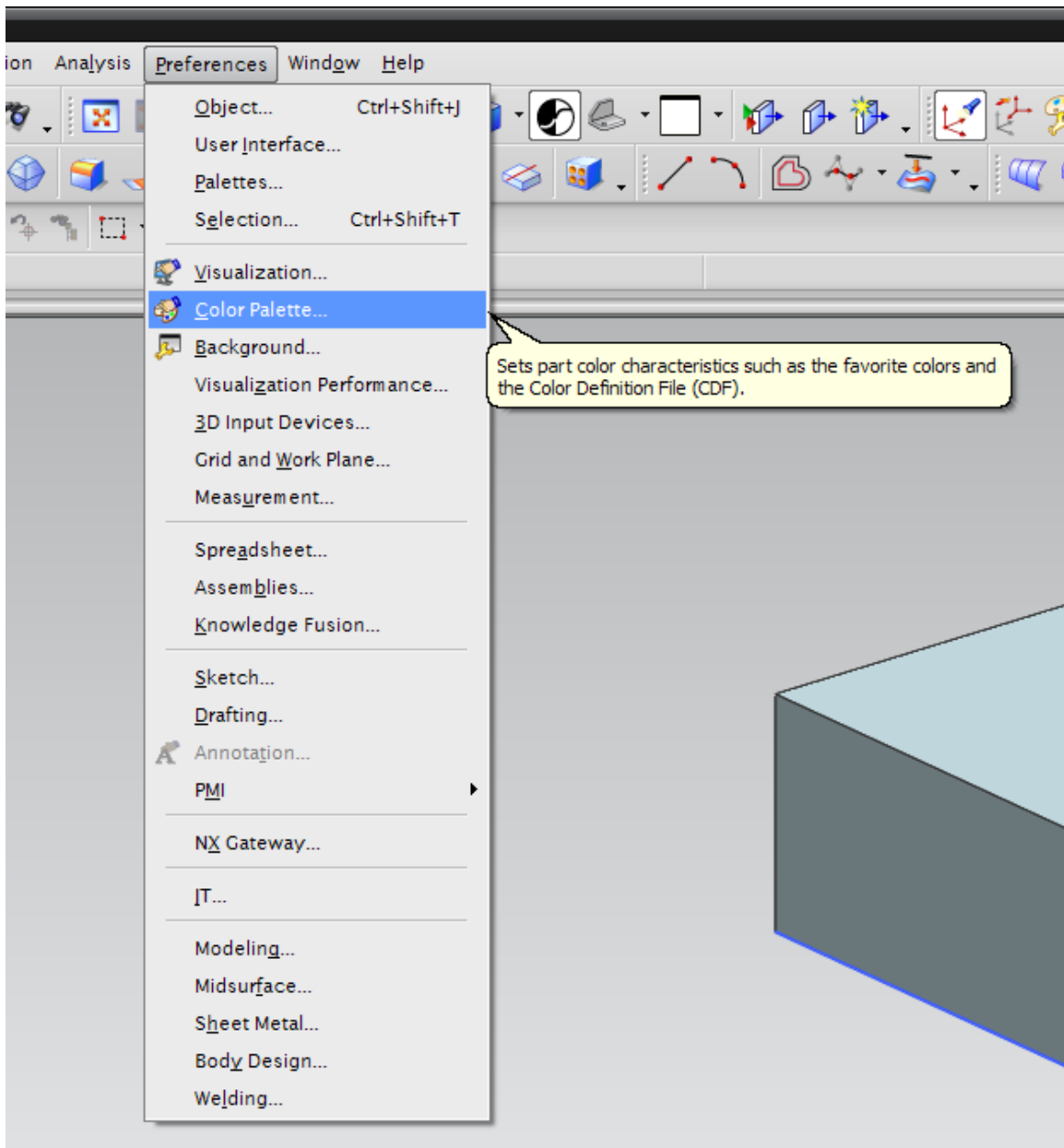
Click the left mouse button to get a 'QuickPick' dialog box as shown in the following figure. This dialog box consists of the list of entities captured within the ball of the cursor. The entities are listed in the dialog with descriptions of the selections in question. Moving the cursor over the options, highlight the corresponding entity in red.



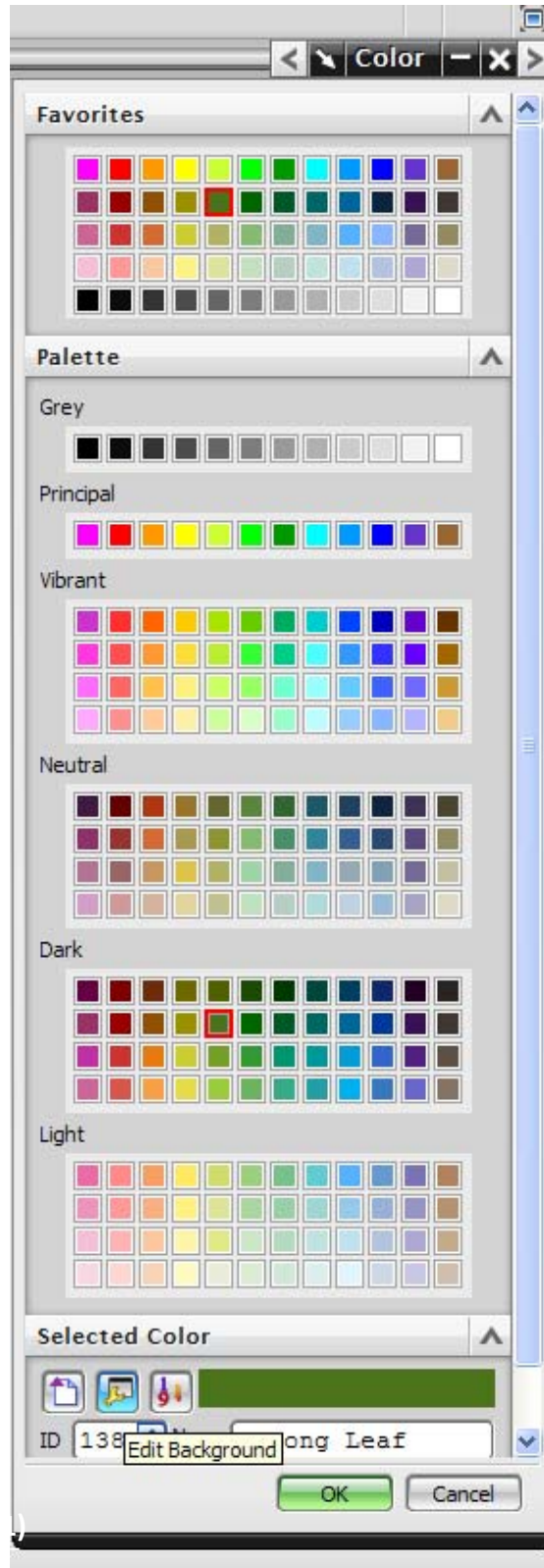
2.3.4

VISUALIZATION (pg 26)

The Colour Palette Tool is not in the visualization preferences dialogue anymore but rather is a separate entity under the preferences menu.

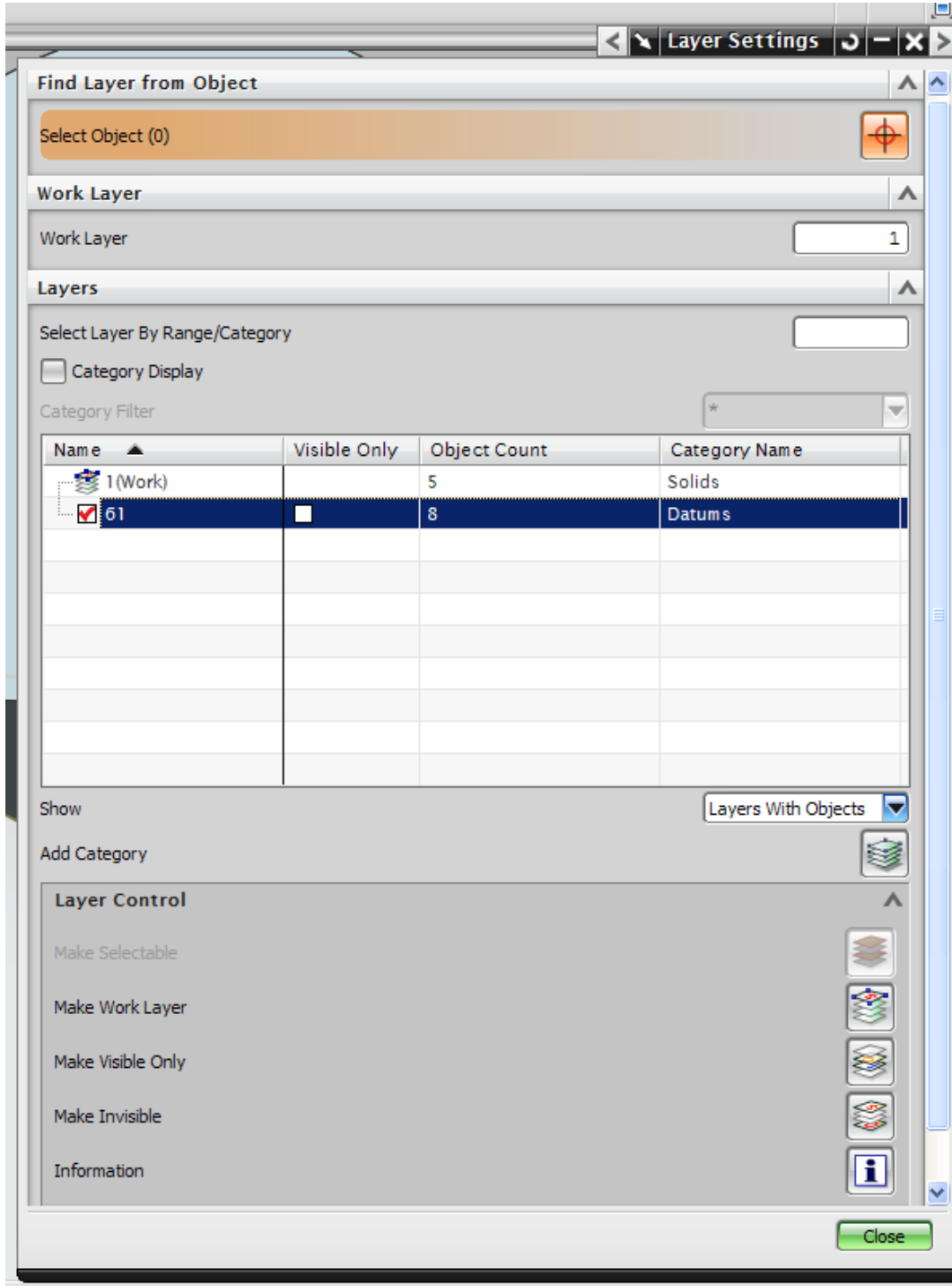


The color palette dialogue window has been updated in presentation:



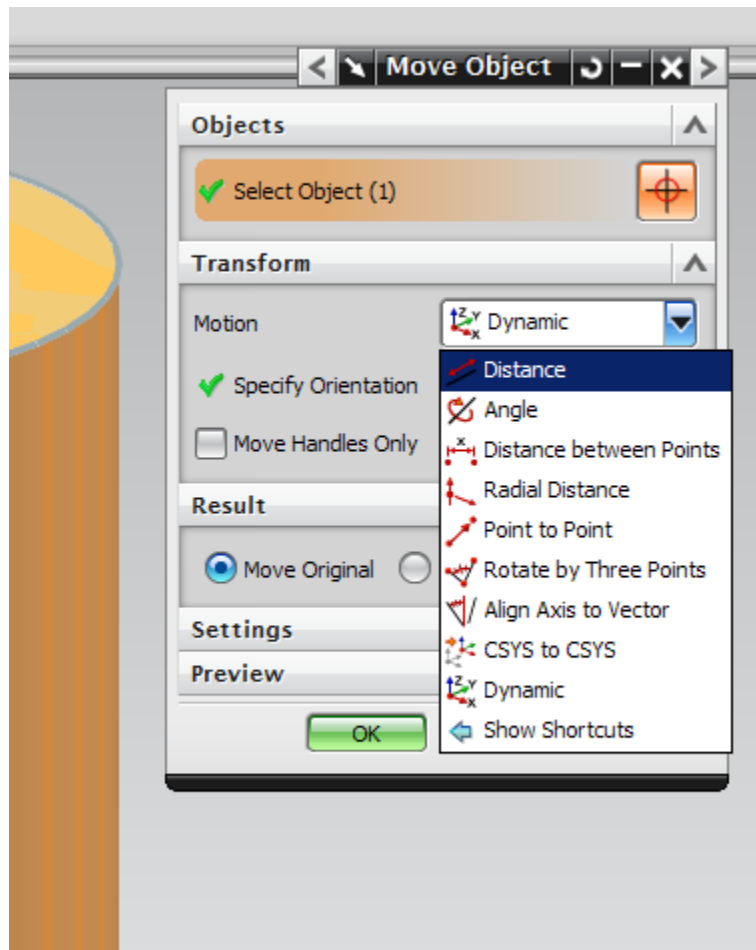
2.5.1 Layer Control (pg 31)

The Layer Settings has the same functionality as in NX5 but has been changed in presentation. Selection options allow toggling of layer functionality.



2.6.2 Transform Functions (pg 38)

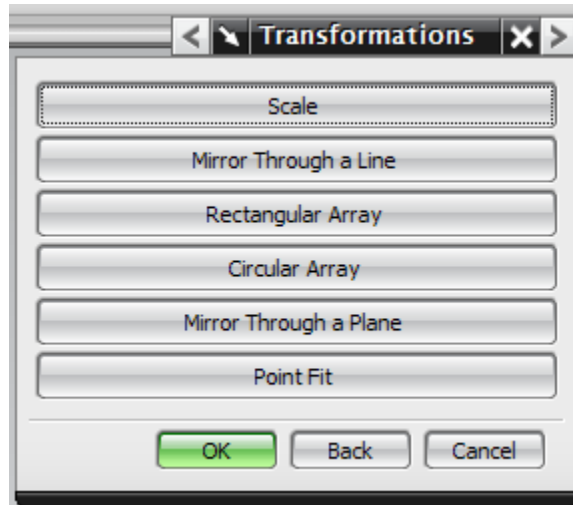
The transform functions can be found under EDIT-->MOVE OBJECT



Here we choose an entity to transform. Transformations allow either moving an object, or copying an object with the new locations parameters. We can:

- Dynamically move objects using handles.
- Move objects a specific distance along a given vector, towards a given vector, or between two points..
- Move and rotate objects along and about a specified vector.

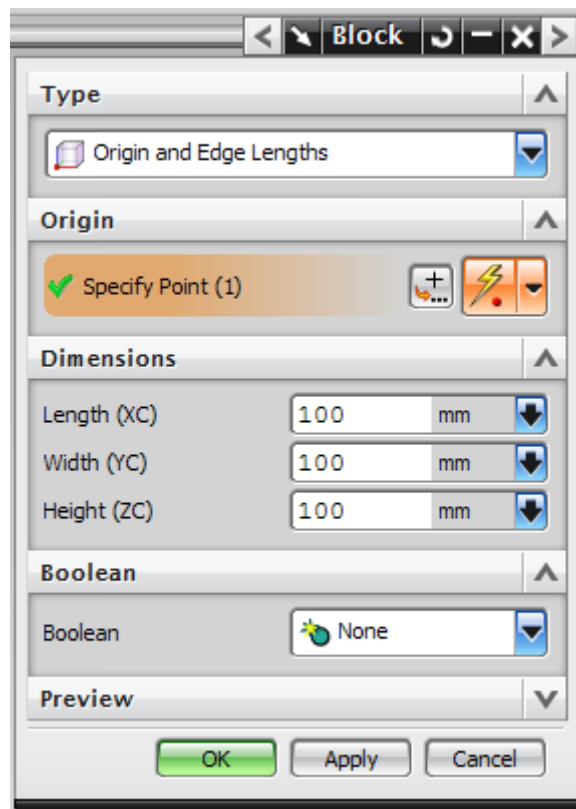
Note: The **Scale**, Mirror, Array and Incremental Dynamics repositioning functions are only available with the **Transform** command on the **Edit** menu. This command is available with the **Essentials with full menus** and the **Advanced with full menus** roles.



3.3.1 Model a Block (pg 44)

Choose INSERT-->DESIGN FEATURE --> BLOCK.

The block window will then appear.

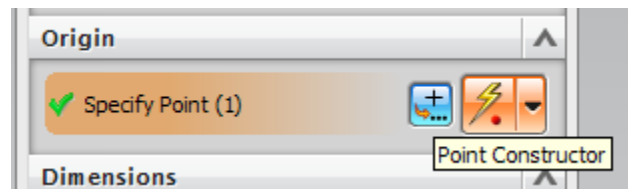


There are 3 ways to create block primitives:

- Origin and Edge Lengths
- Two Points and Heights
- Two Diagonal Points

These options are under the Type section of the Block window in a pull down menu.

- Make sure the Origin, Edge Lengths method is selected.



- Click on the POINT CONSTRUCTOR icon in the Block Window to create the origin.

The drop down menu next to the POINT CONSTRUCTOR contains some commonly used presets.

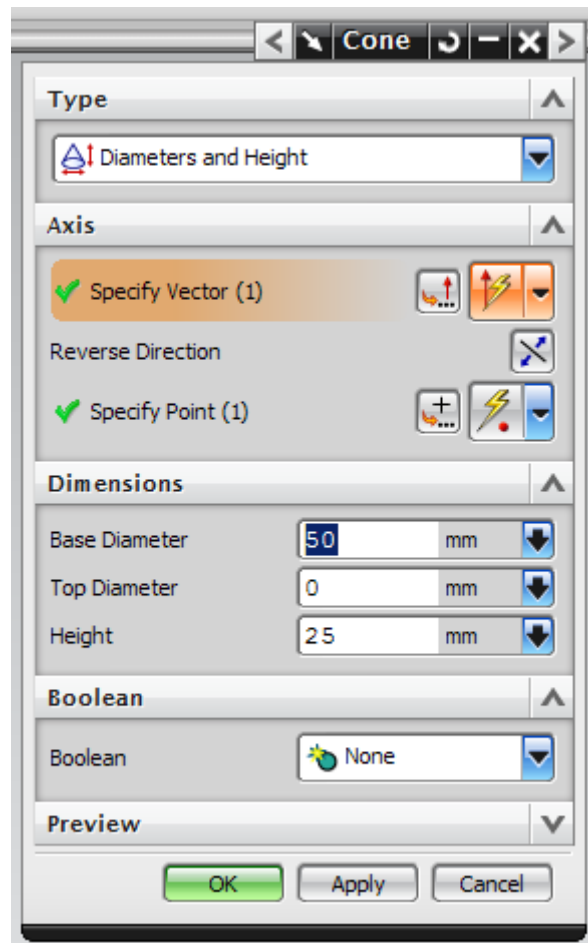
The default should be set to zero for all axis. Select okay.

Select fit and orient view to trimetric to view the newly created feature.

3.3.2 Model a Cylinder (pg 45)

Note changes in interface styles as shown in 3.3.1. All information is accurate in tutorial, except icons and pull down menus are used as illustrated in the above section.

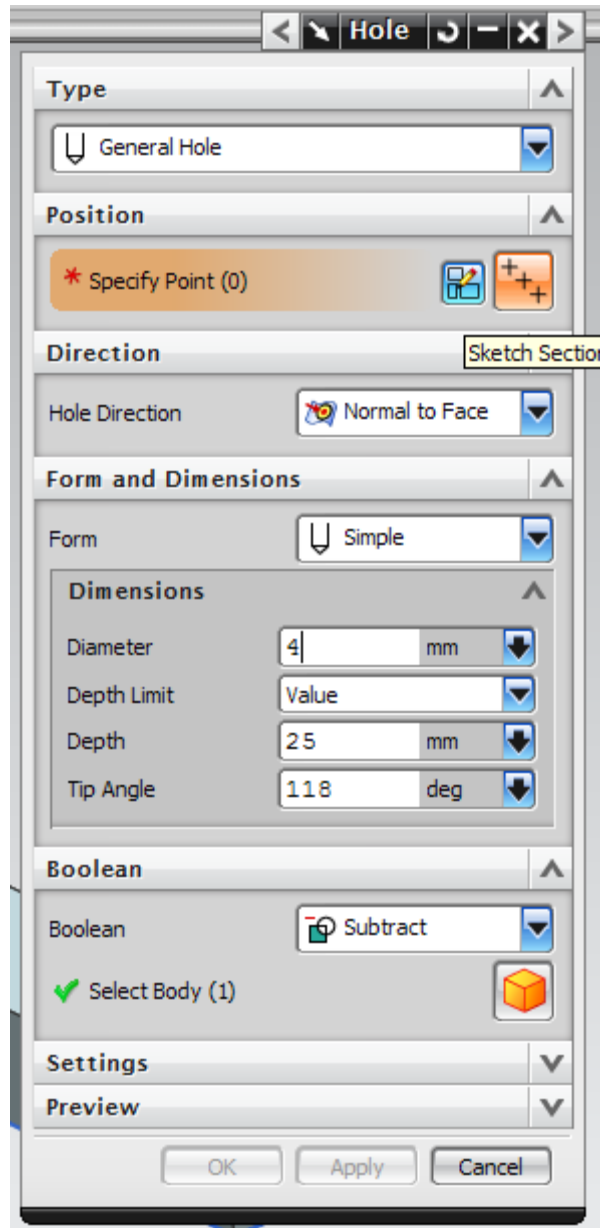
Note that all parameters must be selected within each primitive creation window in order to produce the desired result. Individual construction windows do not pop up as indicated in the tutorial automatically but must be selected in the different sections of the object creation windows. See following figure.



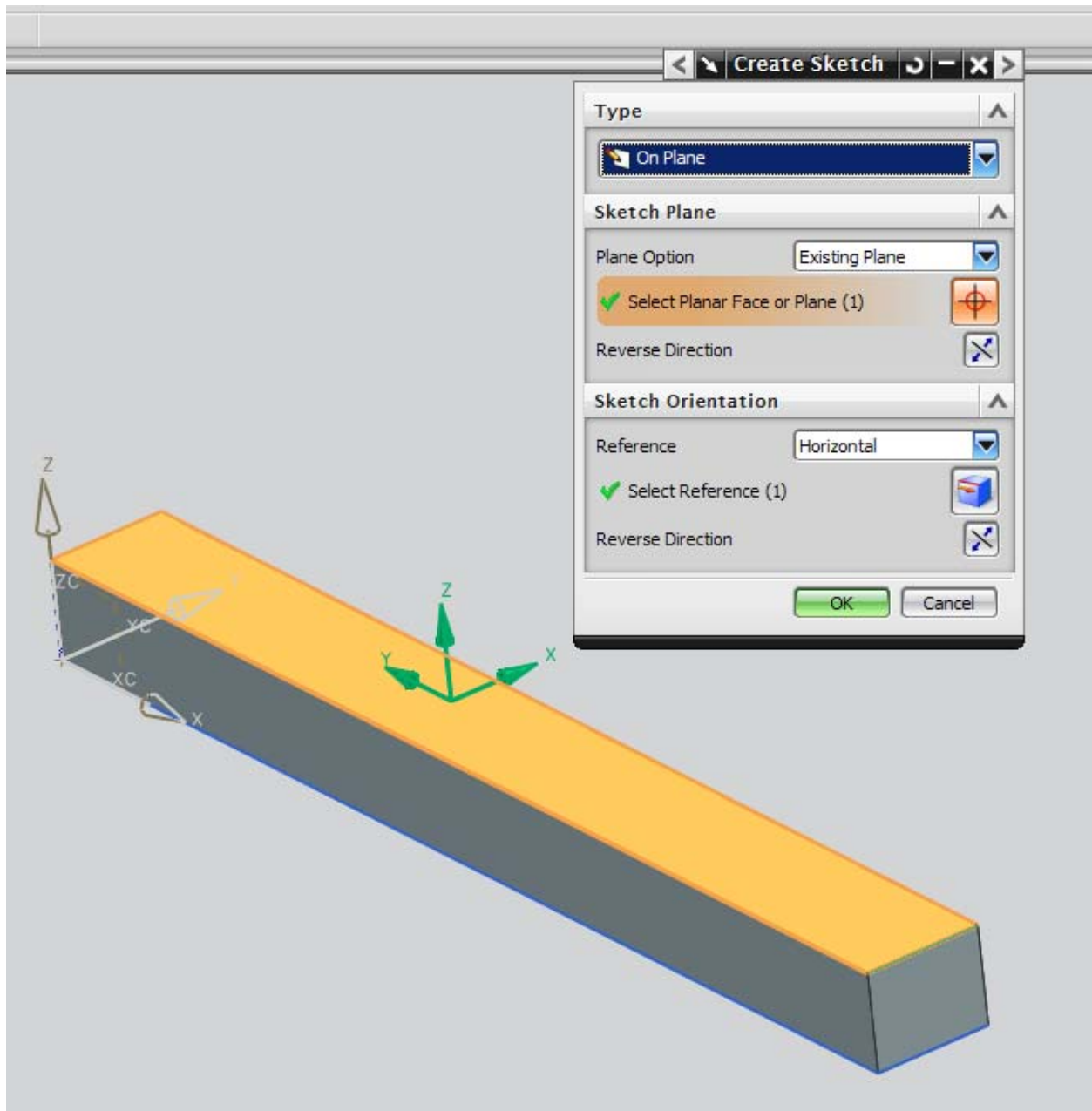
Note the specify Vector call and the Specify Point call.

3.3.2 Remove Features (pg 57)

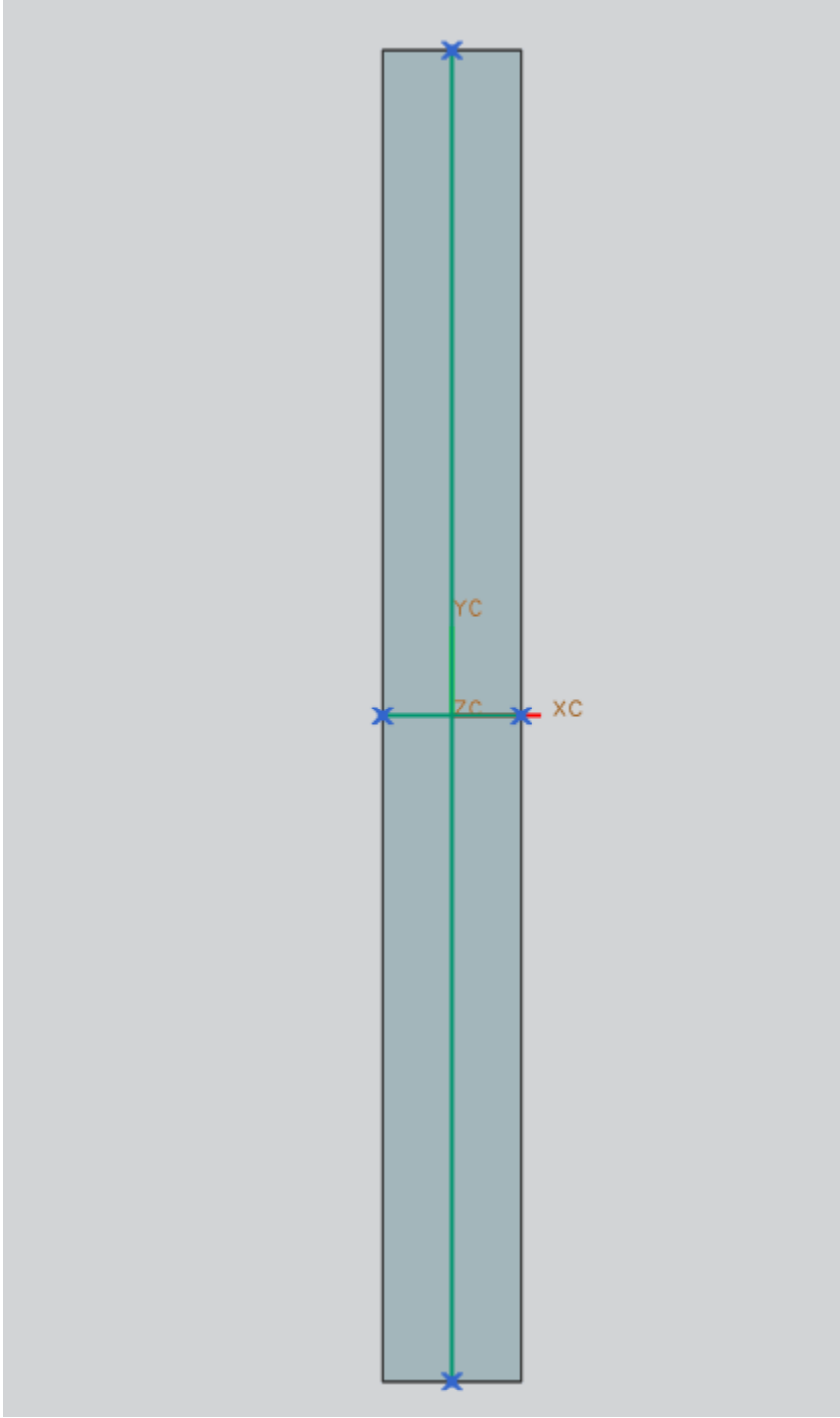
- In the Hole Window, enter the desired hole dimensions including diameter, depth, and tip angle.

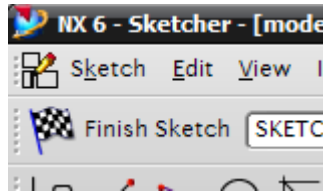


- Selection Criteria may be narrowed by the Hole Direction pull down menu. An existing reference or geometric point may be selected under Position, or the position may be defined by generating a sketch. **click** on the SKETCH SECTION button.

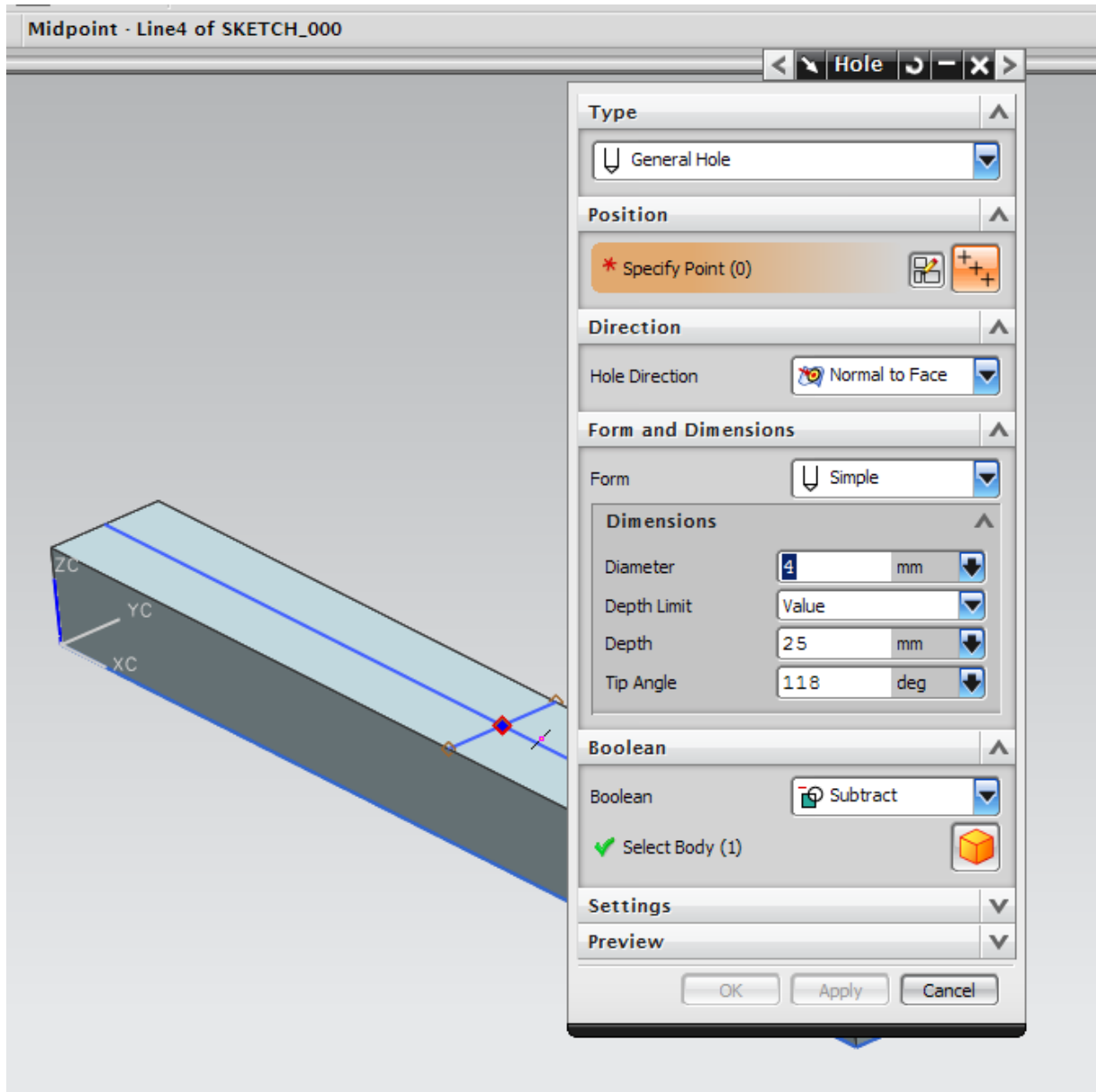


- Selecting the desired face to place the hole will allow placement of the sketch plane. Press Okay in the "Create Sketch" window.
- Draw an appropriate sketch using existing part geometry to construct a point. Press FINISH SKETCH in the top left corner of the menu bar. (See sketching segment of tutorial)





- Select the generated point.



- Press Okay. The hole should appear at the selected point.

NOTE: a reference point could be created from existing geometry using the reference features tools in NX. Then the point could be selected in the Hole window providing an alternative method to producing this feature.

The next lesson of positioning of multiple holes contained in the NX 5 tutorial no longer seems to be a valid way of placing holes on a part. Instead, a pattern of reference points would be generated by either sketch or individually and manually and selected to insert holes. Multiple holes may be inserted at once by selecting more than one point for insertion.

4.2 TYPES OF FEATURE OPERATIONS (pg 61)

Edge blend and Chamfer may be found under INSERT --> DETAIL FEATURE rather than DESIGN FEATURE.

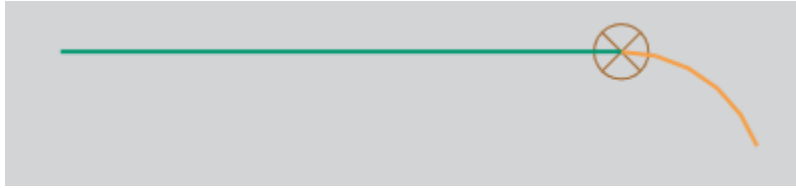
6.2.1.1 SKETCH CURVE TOOLBAR (pg 109)

PROFILE:

The profile tool has an addition for NX6 which allows for a more convenient way of specifying interactions between line segments and arcs:

- start a new sketch in a new model file
- select the profile tool and draw a line segment
- select arc from the profile dialogue
- When mousing over the free end of the profile line segment, you will notice a sub-divided circle appear. This allows specification of alignment between the line segment and the arc end. dragging the pointer through any of the 4 wedges will create an arc that defaults to being aligned to that quadrant relative to the line segment. In this way, tangent arcs and perpendicular arcs may be conveniently produced.

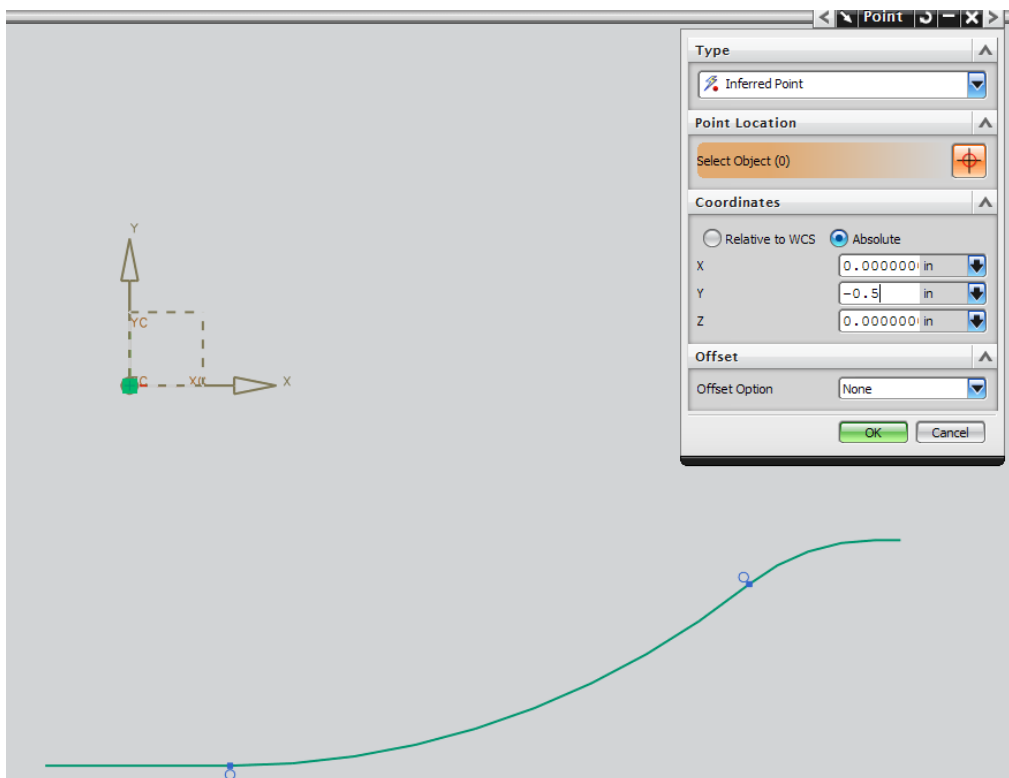


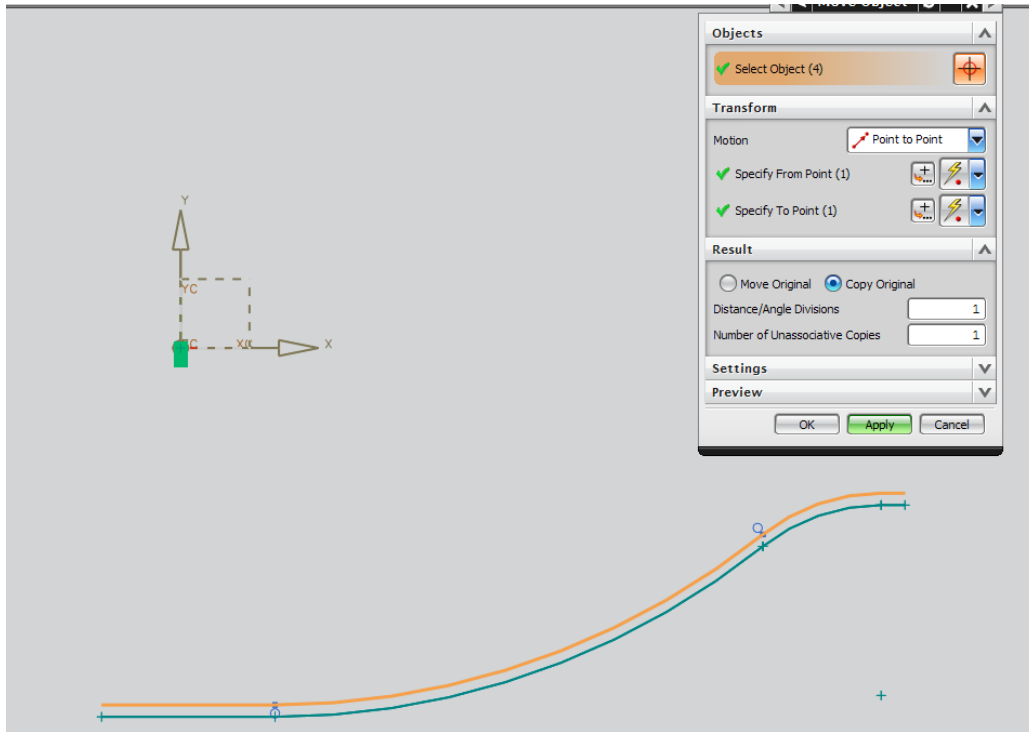


6.2.2 MODEL OF AN IMPELLER LOWER CASING (pg 121)

As noted above, some of the transform functions are now under the "move objects" tool in the edit menu:

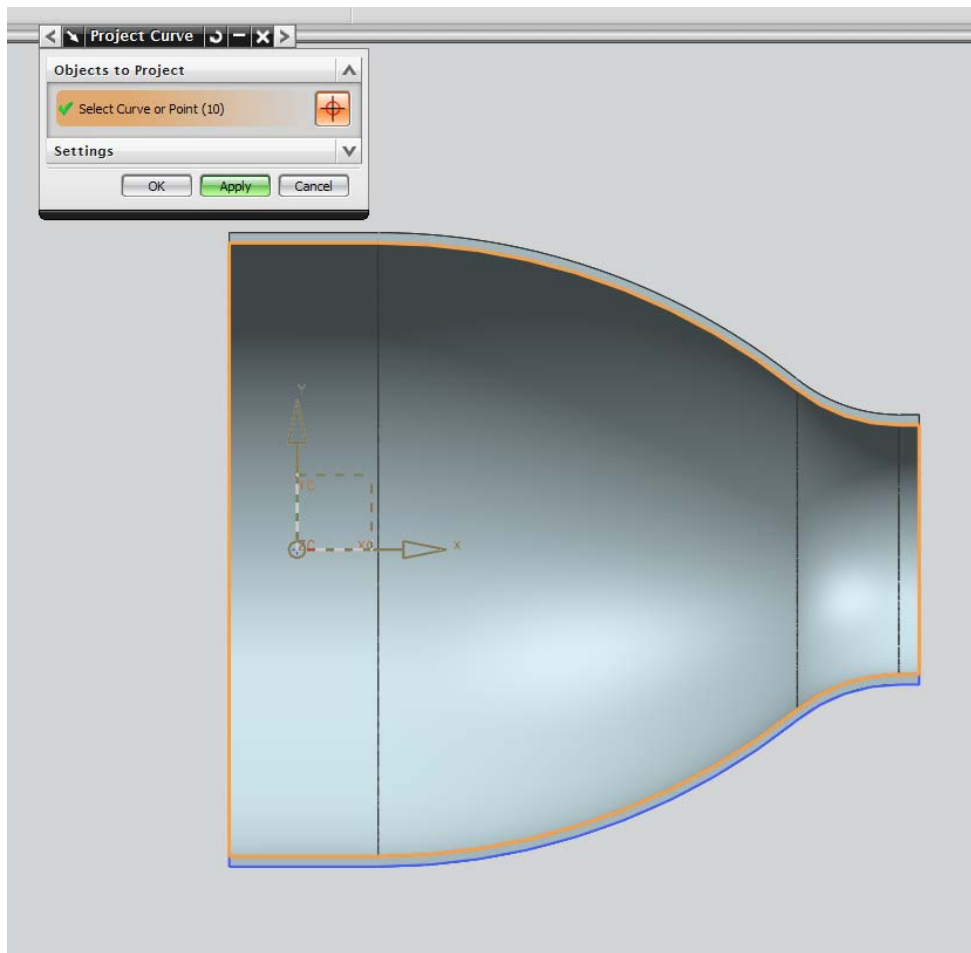
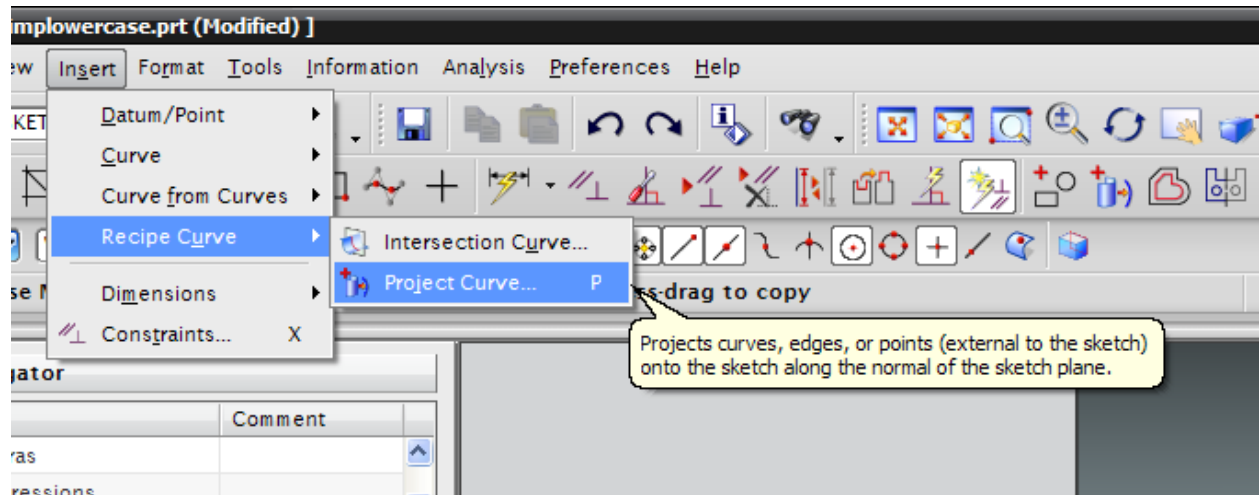
- Choose EDIT-->MOVE OBJECT and select the COPY ORIGINAL option
- Select the vertex at the coordinate reference system as the first point and enter (0, 0-0.5, 0) into the point constructor for the second point.
- select all the objects in the curve and press okay.

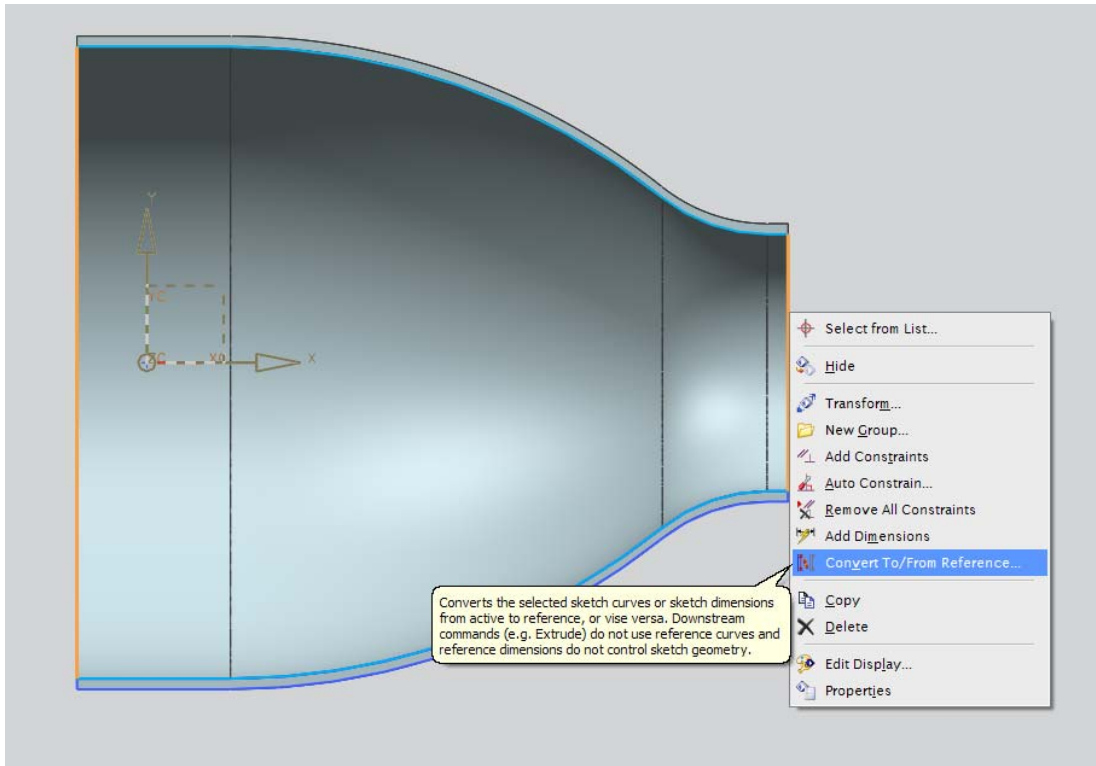




For creating the edge of the part, a different set of steps is required. The functionality of the move objects tool set seems to be limited in that it will not allow copying of entities from one sketch into another. To overcome this, model geometry will be converted instead.

- Select INSERT-->RECIPE CURVE-->PROJECT CURVE
- Select the interior surface of the revolved shape and press OK
- Select all shapes not to be used in the edge extrusion and make them reference geometry by RIGHT CLICKING and pressing MAKE REFERENCE GEOMETRY.
- Translate/copy outer curve as described in above process to obtain desired results.





When copying the edge to the other side, select the Y-C plane manually while the inferred option is selected, and press OKAY and COPY.

