To Create a Tool Model

1. Create a new Pro/ENGINEER model of type Part or Assembly, give it the name of the tool.
2. Reproduce the tool geometry by using the appropriate construction features (protrusions, cuts, and so on).
3. Create a coordinate system to represent the tool origin, that is, the tool control point. This is the point that will follow the tool path computed for the NC sequence. Make sure the Z-axis of the coordinate system points in the upward direction (into the tool) for Milling and Holemaking tools; for Turning, the axes of the tool coordinate system must be oriented so that they coincide with the direction of the NC sequence coordinate system's axes when the tool is in default orientation. Change the coordinate system's name to `TIP`.

   3. Establish associativity between the dimensions of the model and the tool parameters. There are two ways to do this:

   - Modify appropriate dimension symbols so that they correspond exactly with the parameter names. Select the feature, display its dimensions, right-click the dimension text, and click **Properties** on the shortcut menu. The **Dimension Properties** dialog box opens. Go to the **Dimension Text** tabbed page and type the new symbolic name in the **Name** box, for example, `Cutter_Diam`.
   - Add parameters to the model with the names corresponding exactly with the tool parameter names. This method is convenient when you want to define the tool parameters directly in the tool assembly (for example, `Cutter_Diam` for an insert drill as opposed to a drill bit).

   Notes:

   - Parameter names are case-insensitive. For example, when modifying a dimension symbol or adding a model parameter for Cutter_Diam, you can use `Cutter_Diam`, `cutter_diam`, or `CUTTER_DIAM`; NC Manufacturing recognizes either of these strings as a tool parameter name.
   - If an assembly is to be used as a tool model, you can modify dimension symbols or add parameters to any of the component parts as well as the assembly itself.

See Also

About Solid Tool Models

Example: A Solid Tool Model for Milling

To Assign Tool Material and Number of Teeth