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described in the sketches and features, can be used to quickly modify/update the design. In Creo Parametric, the parametric part modeling process involves the following steps: 1. Set up Units and Basic Datum Geometry. 2. Determine the type of the base feature, the first solid feature, of the design.

Parametric Modeling with Creo Parametric 2

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Entering Sketcher Directly from Creo Parametric Select the NEW FILE icon from the ribbon at the top of the screen. Save as >File>New. When the New file window opens, select Sketch as the type, then enter a valid filename without the extension. Pick OK. Figure 3-2 New Sketch Named

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PTC Creo - Wikipedia

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