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Figure 3-2 Select the Front and Up Direction. Select the Owner of the view. Select the Front direction by Clicking the Front Dir button and

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then clicking the face, an edge or a pair of points to define the front direction. The arrow will point into the view.

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Creo2.0 This video is about how to prepare Thor Hammer in Creo 2.0
parametric with details steps and explanation. Command used:-
Extrude, palette, sweep, reference, mirror,, etc.

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PTC Creo Parametric | GrabCAD Tutorials

Creo Elements/Pro, or Creo Elements/Direct Modeling, formerly known, together with Creo Parametric, as Pro/Engineer and Wildfire, is a solid modeling or CAD, CAM, CAE, and associative 3D modeling application, running on Microsoft Windows. It is the direct modeling application of a suite of 10 that provide collaborative solid modeling, assembly modeling, 2D orthographic views, finite element analysis, parametric modeling, sub-divisional and NURBS surface modeling, direct drafting, and NC and tool

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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. Note that Extrude, Revolve, or Sweep operations are the most common types of base features. 3.

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