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In Pro/ENGINEER, 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 Sweepoperations are the most common types of base features. 3.

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Complete installation of required release of Pro/ENGINEER. Extract the dataset archive contents to a

New Folder. - preserve folder structure - a simple path with no spaces is recommended (e.g.,

D:\PTC\WF4_Tutorial) Edit startpro.ini to specify Pro/ENGINEER install path (PROE_DIR). - you will be prompted if this value is missing or incorrect