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A Beginner's Guide to Pro/ENGINEER (version 2000i) Tom Chase September 7, 2000 Contents 1 Starting and Exiting Pro / ENGI-NEER 2 2 Obtaining Further Information 4 3 Creating a Base Feature

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What kinds of help tools (online tutorials/printed manuals/general help) are provided with the Pro/ENGINEER 2001 Student Edition? PTC's Online Help is shipped with Pro/ENGINEER 2001 Student Edition and is installed by default. The Online Help can be started from within Pro/ENGINEER by selecting #Help, #Pro/E Help System.

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Pro/ENGINEER 2000i [Gary Lamit] on Amazon.com. \*FREE\* shipping on qualifying offers. Lamit's learn-by-doing approach to using Pro/Engineer 2000i instructs readers in how to use Pro/Engineer software in twenty-two lessons. Project-based lessons incrementally introduce users to a new set of commands and concepts that are applied to parts

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Pro/ENGINEER Tutorial (Release 20/2000i) (A Click by Click Primer) and MultiMedia CD [Roger Toogood] on Amazon.com. \*FREE\* shipping on qualifying offers. Book by Toogood, Roger

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Pro/Engineer WildFire 5 is a powerful 3D modeling tool by PTC with all the necessary features and options. This free download is the standalone setup of Pro/Engineer for Windows 32-bit and 64-bit.

### ~~Pro Engineer Tutorial Ver. 2000i2 (sample)~~

And I had already been using Pro/E for a few years. Trying to learn them both will really make life difficult. If you are working at a company still using 2000i (the only reason that I could see you wanting to learn it), they might have some old manuals (did they stop printing them by then?) or training guides that might help you.

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To publish eDrawings you must have either Pro/ENGINEER 2000i 2, Pro/ENGINEER 2001, Pro/ENGINEER Wildfire or Pro/ENGINEER Wildfire 2.0, SolidWorks 2003 or higher, AutoCAD 2000, or AutoCAD R14.x. Is technical support offered for eDrawings? No, there is no technical support offered for the free version of eDrawings for Pro/ENGINEER.

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Pro/ENGINEER (commonly referred to as Pro/E or ProE) is a 3D CAD parametric feature solid modeling software created by Parametric Technology Corporation (PTC). Its direct competitor's software are SolidWorks, Inventor, SolidEdge, Unigraphics, and CATIA. It runs on Microsoft Windows and provides solid modeling, assembly modelling and drafting functionality for mechanical engineers.

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The name changed to Creo 1.0 after Pro/ENGINEER Wildfire 5.0 (rebranded PTC Creo Elements/Pro), took place on October 28, 2010, which coincided with PTC's announcement of Creo, a new design software application suite. For the first 10 years, PTC generally released 2 versions per year, with some exceptions. The initial release (Rev 1) was in 1988.

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