

Pro Engineer Wildfire 50 Tutorial

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5.0 out of 5 stars Pro/Engineer Wildfire 5.0 Tutorial with CD. Reviewed in the United States on June 23, 2013. Verified Purchase. This book is everything I expected, contents and CD. Great reference book for someone who hasn't used Pro for awhile and needs to catch up to the latest version. One person found this helpful.

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1-4 Parametric Modeling with Pro/ENGINEER 3. In the New dialog box, confirm the model's Type is set to Part (Solid Sub- type). 4. Enter Adjuster as the part Name as shown in the figure. 5. Turn off the Use default template option. 6. Click on the OK button to accept the settings. 7.

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Pro/E Wildfire. Oh, and if you're running Wildfire and you open up a file that you created using Pro/E 2001, Wildfire corrupts the file so that you can't open it in 2001 anymore. You're locked into using Wildfire, so essentially you can't get a refund for the program once you've used it... unless you're OK with losing your work.

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