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Dec 08, 2009 Introductory level and tutorial style Covers Pro/ENGINEER and parametric modeling with hands on tutorials New in this edition is coverage of Pro

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This tutorial shows Datum Target Annotation is an enhancement of Pro/ENGINEER Wildfire 5.0, Locate HIDDEN Components in Model Tree; PTC Creo Elements/Pro & Pro

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Chapter. 14 Surface Modeling Learning Objectives After completing this chapter you will be able to: Creating an Extruded Surface. Creating a Revolved Surface.

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