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This is the first in our series of 6 tutorials on Pro/CABLING. This powerful module will allow you to rapidly create and route wires and harnesses through Pr...

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A Conductor is an internal current carrying entity within a cable. What is a Spool? A Spool is used to determine the majority of parameters, which describe a wire or cable. It is analogous to a reel of wire or cable from which individual wires or cables are cut. One spool can provide for only one type of wire or cable, but multiple wires or ...

INTRODUCTION TO Pro/CABLING™ Pro/ENGINEER Wildfire 2

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Cabling - Creo Elements/Pro 5.0

Getting Started with Pro/ENGINEER Wildfire is a tutorial-based introduction to creating parts, assemblies and drawings in Pro/ENGINEER. If you follow the complete series of procedures, you will learn how Pro/ENGINEER passes 3D design information to and from every design stage, from solid part creation, to part assembly, to the

Getting Started with Pro/ENGINEER Wildfire 4

The Path to Creo. Built on the legacy of Pro/ENGINEER, CoCreate and ProductView, Creo is a family of design software which will help companies unlock potential within their organizations. Product designers and engineers will be more productive, enabling better data sharing and design reviews with customers and suppliers, and preventing unforeseen service and manufacturing issues.

Pro/ENGINEER | PTC

Cabling Design Routing Cables Tips and Tricks. This tutorial is part of a course. View the full course. Views 5937 Views Difficulty level Introductory Jim Barrett-Smith, PTC. In this tutorial you will learn how to filter logical cables, add locations to axis and understand how to recognize in which wire ...

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Re: Pro/Engineer Wire Harness Tutorial Thanks for the positive feedback brian, i would much rather use Pro/Cabling, but that is an extra option that i dont have at home. 0 Kudos

Pro/Engineer Wire Harness Tutorial - PTC Community

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Start Pro/ENGINEER Wildfire 3.0 if necessary. If Pro/ENGINEER is already running ensure all windows are closed, and all items from the previous exercise are erased from memory. In the Navigator Folder Browser, browse to the following folder C:\HANDS-ON_WF3\DRAWING_TUTORIAL. Right-click the DRAWING_TUTORIAL folder and select Set Working Directory.

Introduction to Creating Drawings in Pro/Engineer

Elements/Pro & Pro/ENGINEER 5.0; This short tutorial gets you started creating a sphere with Pro/ENGINEER. To create spheres you can use the Revolve feature. A revolved feature is an axis symmetric shape that is created by revolving an open or closed section view by a specified angle around a central axis.

Creo Elements/Pro & Pro/ENGINEER - PTC Learning Connector

Cable (called Harness) parts can not exist without main physical assembly. It is a simple box part to hold for Connectors. One Dual Connector with create two coordinate system at the shown location (Z-Direction pointing outward) for entry points for wire.

Cable Design with Pro-Engineer Wild Fire 2 (Part-1 Manual ...

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