



MANUAL DE: UNIGRAPHICS NX



Elaborado por: Cutting Edge, S.L.

Unigraphics Nx Student Manual

Tutorial Books



Unigraphics Nx Student Manual:

Practical Applications of Unigraphics Unigraphics Solutions Inc,2000 *The Hindu Index* ,2003 *Annual Index/abstracts of SAE Technical Papers* ,2006 **Nx 11 Tutorial** Tutorial Books,2016-10-05 NX 11 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 11 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial Download Resource files from <http://onlineinstructor.org/book/nx11tutorial> **Nx 11 Tutorial** Online Instructor,2016-10-08 NX 11 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 11 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial Download Resource files from <http://onlineinstructor.org/book/nx11tutorial> **NX 12 Tutorial** Online Instructor,2020-11-16 NX 12 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 12 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations simulation basics product manufacturing information and rendering **Unigraphics NX for Total Product Engineering** Unigraphics Solutions Inc,2002 *Micrografx Designer* Micrografx, Inc,1992 [Nx 10 Tutorial](#) Online Instructor,2015-03-31 NX 10 Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling techniques The Author guides readers through NX 10 with clear and step by step tutorials that help you to design solid models from day one The first four chapters of this book cover the user interface part modeling assemblies and drawings After learning the basics you can learn additional sketching tools feature modeling tools expressions sheet metal modeling some advanced assembly techniques drawing annotations and simulation basics Table of Contents 1 Getting Started 2 Modeling Basics 3 Constructing Assembly 4 Generating Drawings 5 Sketching 6 Additional Modeling Tools 7 Top

Down Assembly 8 Dimensions and Annotations 9 Simulation Hands on Tutorial **Student Manual and Data Course**
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 Books,2019-11-27 NX Tutorial is written to help new users to learn the basics of NX and some advanced solid modeling
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 Assemble create pro drawings Explore Sheet Metal Surface Design Realize Shape Unleash creativity NX journey begins
Nx 10 for Beginners Tutorial Books,2015-08-16 NX 10 For Beginners introduces you to the basics of NX 10 by using step
 by step instructions You begin with brief introduction to NX 10 and the User Interface ribbon environments commands and
 various options Within a short time you will learn to create 2D sketches that form the basis for 3D models You will learn to
 sketch on three different planes Front Top and Right planes You will use various sketching tools such as line rectangle circle
 and so on You will also learn to modify sketches using tools such as trim extend fillets and so on Learn to use geometric
 constraints and dimensions to achieve a definite shape and size of the sketch Sketches are converted into 3D features such as
 Extrude Revolve and so on You combine or subtract features to achieve the final part You can also add placed features sketch
 less features such as Fillets and Holes to the 3D geometry You explore mirroring and patterning commands to create
 repetitive features You will learn to use some additional modeling tools and work with multi body parts Learn to modify part
 geometry by editing sketches and feature parameters You explore Synchronous Modeling tools to modify the Part geometry
 by modifying its faces You build assemblies after creating parts There are two methods to build assemblies Bottom up and
 Top down In the Bottom up method you bring all the parts together and add constraints between them In the Top down
 method you create parts in the assembly level You explode assemblies to show the manner in which they were assembled You
 create Drawings of the parts and assemblies You insert part views and add dimensions and annotations to complete the
 drawing In case of assembly drawings you insert assembly views add Bill of Materials Balloons and Revision table The Sheet
 Metal design chapter covers various tools used to build sheet metal parts from scratch You will also learn to convert an
 existing part geometry into sheet metal part You also create flat patterns and 2D sheet metal drawings Finally you explore
 the surface modeling tools used to create complex shapes Table of Contents 1 Getting Started with NX 10 2 Sketch
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