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use. We first released the tutorial for Unigraphics 18 and later updated for NX2 followed by the updates for NX3, NX5 and NX7. This write-up further updates to NX9.0. Our previous efforts to prepare the NX self-guiding

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tutorial were funded by the
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Software - Class Finder Important

and significant version changes

between Unigraphics NX release

5.0 and 6.0 that may cause

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frustration to users attempting to complete the tutorial for NX6 are
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Technologies

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Unigraphics NX is a cost efficient

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CAD software product developed and marketed by UGS and is used by many companies for designing and manufacturing products which varies everywhere from printing presses to locomotives, Lego bricks to International Space Station, cars and trucks to fighter

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Station, cars and trucks to fighter planes, cameras and razors to steam turbines and everything in between. Unigraphics Guide - modapktown.com Beginners Guide to Unigraphics. Unigraphics NX is a cost efficient CAD Page 8/26

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UG NX Tutorial : From Beginner to Advanced | Smart Guide NX, formerly known as "unigraphics", is an advanced high-end CAD/CAM/CAE, which has been

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owned since 2007 by Siemens PLM Software. In 2000, Unigraphics purchased SDRC I-DEAS and began an effort to integrate

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Unigraphics NX is used in training
to learn the development process
for computer aided design (CAD)
and computer aided

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manufacturing (CAM). Product lifecycle management (PLM) is an information management system that can integrate data, processes, business systems and, ultimately, people in an extended enterprise.

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users of NX2 software who want to learn professional techniques which will help them work smarter and faster. This Unigraphics book covers advanced topics in modeling and assemblies, and special techniques required to render

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geometry from various industries. The book specifically focuses on complex surfacing techniques, sheet metal parts, advanced assemblies and advanced techniques to design consumer products and injection molded parts. Content for this Advanced

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NX2 book is based on requests we received from numerous readers of our popular basic books, Practical Unigraphics NX and NX2 Modeling for Engineers. Like our training classes, this book is project-oriented. The exercises provided in this book

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are classroom tested, and are guaranteed to give you the knowledge you need to learn advanced NX2 techniques.

Practical Unigraphics NX2

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Modeling for Engineers is a cost-effective, self-paced course in UGS NX2 software. The NX2 book includes practical exercises, self-tests, and timesaving tips that are applicable for both NX and NX2. This Unigraphics book is a joint effort by Design Visionaries

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to bring to life DV President Stephen Samuel's vision of compiling and publishing the NX training exercises that he has been creating for the engineering community for years. Like his Unigraphics training programs, this book is also project-

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oriented. Methods outlined in this UG book go beyond an academic use of Unigraphics-they are tricks of the trade that come from thousands of hours of actual use of Unigraphics to design some of the most difficult products in the world. In many cases, the

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examples and exercises emulate actual design work. The exercises provided in this UG book are classroom tested, and are guaranteed to give you the knowledge you need to learn NX2.

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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

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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

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tools, expressions, sheet metal modeling, some advanced assembly techniques, drawing annotations, simulation basics, PMI, and rendering.

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The primary goal of Parametric Modeling with Siemens NX is to introduce the aspects of designing with Solid Modeling and Parametric Modeling. This text is intended to be used as a practical training guide for students and professionals. This text uses

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Siemens NX as the modeling tool, and the chapters proceed in a pedagogical fashion to guide you from constructing basic solid models to building intelligent mechanical designs, creating multi-view drawings and assembly models. This text takes

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a hands-on, exercise-intensive approach to all the important Parametric Modeling techniques and concepts. This textbook contains a series of fifteen tutorial style lessons designed to introduce beginning CAD users to NX. This text is also helpful to NX

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users upgrading from a previous release of the software. The solid modeling techniques and concepts discussed in this text are also applicable to other parametric feature-based CAD packages. The basic premise of this book is that the more designs

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you create using NX, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. This book does not attempt to cover all of NX's features, only to provide an

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introduction to the software. It is intended to help you establish a good basis for exploring and growing in the exciting field of Computer Aided Engineering. This book also introduces you to the general principles of 3D printing including a brief history of 3D

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printing, the types of 3D printing technologies, commonly used filaments, and the basic procedure for printing a 3D model. 3D printing makes it easier than ever for anyone to start turning their designs into physical objects, and by the end

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of this book you will be ready to start printing out your own designs.

This textbook explains how to create solid models, assemblies and drawings using Siemens NX 12. NX is a three dimensional

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CAD/CAM/CAE software developed by Siemens PLM Software Inc., Germany. This textbook is based on NX 12. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our web- site. Almost all files are

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in NX 6.0 so readers can open the files using NX 6.0 and later releases. It is assumed that readers of this textbook have no prior experience in using Siemens NX for modeling 3D parts. This textbook is suitable for anyone interested in learning 3D

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modeling using Siemens NX. Each chapter deals with the major functions of creating 3D features using simple examples and step by step, self-paced exercises. Additional drawings of 3D parts are provided at the end of each chapter for further self exercises.

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The final exercises are expected to be completed by readers who have fully understood the content and completed the exercises in each chapter. Topics covered in this textbook - Chapter 1: Basic components of Siemens NX 12, options and mouse operations. -

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Chapter 2: Basic step by step modeling process of NX 12. - Chapter 3 and 4: Creating sketches and sketch based features. - Chapter 5: Usage of datums to create complex 3D geometry. - Chapter 6: Additional modeling commands such as

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fillet, chamfer, draft and shell. -
Chapter 7: Modification of 3D
parts to take advantage of
parametric modeling concepts. -
Chapter 8: Copying features,
modeling objects and bodies. -
Chapter 9: Additional modeling
commands such as trim body,

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tube, sweep along guide, emboss and various commands in synchronous modeling. - Chapter 10: Advanced sketch commands. - Chapter 11: Measuring and verifying 3D geometries. - Chapter 12 and 13: Constructing assembly structures and creating

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or modifying 3D parts in the context of assembly. - Chapter 14 and 15: Creating drawings for parts or assemblies. - Appendix A: Selecting Objects

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