
Unigraphics Nx5 Tutorial

When somebody should go to the ebook stores, search foundation by shop, shelf by shelf, it is in fact problematic. This is why we offer the book compilations in this website. It will definitely ease you to see guide **Unigraphics Nx5 Tutorial** as you such as.

By searching the title, publisher, or authors of guide you essentially want, you can discover them rapidly. In the house, workplace, or perhaps in your method can be all best place within net connections. If you take aim to download and install the Unigraphics Nx5 Tutorial, it is extremely easy then, back currently we extend the colleague to buy and create bargains to download and install Unigraphics Nx5 Tutorial as a result simple!

*Unigraphics
Nx5 Tutorial*

2022-02-28

RICHARD NOVAK

NX 12 Tutorial Infinity

Publishing
CATIA V5-6R2017 for
Designers is a
comprehensive book
written with the intention

of helping the readers
effectively use all solid
modeling tools and other
features of CATIA
V5-6R2017. This book

provides elaborate and clear explanation of tools of all commonly used workbenches of CATIA V5-6R2017. After reading this book, you will be able to create, assemble, and draft models. The chapter on the DMU Kinematics workbench will enable the users to create, edit, simulate, and analyze different mechanisms dynamically. The chapter on Generative Shape Design explains the concept of hybrid designing of models. Also, it enable the users to quickly model both simple

and complex shapes using wireframe, volume and surface features. The chapter on the FreeStyle workbench will enable the users to dynamically design and manipulate surfaces. In this book, a chapter on FEA and structural analysis has been added to help users to analyze their own designs by calculating stresses and displacements using various tools available in the Advanced Meshing Tools and Generative Structural Analysis workbenches of CATIA

V5-6R2017. The book explains the concepts through real-world examples and the tutorials used in this book. After reading this book, the users will be able to create solid parts, sheet metal parts, assemblies, weldments, drawing views with bill of materials, presentation views to animate the assemblies, analyze their own designs and apply direct modeling techniques to facilitate rapid design prototyping. Also, the users will learn the editing techniques that are essential for

making a successful design. Salient Features Consists of 19 chapters that are organized in a pedagogical sequence. Detailed explanation of CATIA V5-6R2017 tools. First page summarizes the topics covered in the chapter. Hundreds of illustrations and comprehensive coverage of CATIA V5-6R2017 concepts and techniques. Step-by-step instructions that guide the users through the learning process. More than 40 real-world mechanical engineering designs as

tutorials and projects. Technical support by contacting techsupport@cadcim.com. Additional learning resources at <https://allaboutcadcam.blogspot.com> Table of Contents Chapter 1: Introduction to CATIA V5-6R2017 Chapter 2: Drawing Sketches in the Sketcher Workbench-I Chapter 3: Drawing Sketches in the Sketcher Workbench-II Chapter 4: Constraining Sketches and Creating Base Features Chapter 5: Reference Elements and

Sketch-Based Features Chapter 6: Creating Dress-Up and Hole Features Chapter 7: Editing Features Chapter 8: Transformation Features and Advanced Modeling Tools-I Chapter 9: Advanced Modeling Tools-II Chapter 10: Working with the Wireframe and Surface Design Workbench Chapter 11: Editing and Modifying Surfaces Chapter 12: Assembly Modeling Chapter 13: Working with the Drafting Workbench-I Chapter 14: Working with the Drafting

Workbench-II Chapter 15:
Working with the Sheet
Metal Components
Chapter 16: DMU
Kinematics Chapter 17:
Introduction to Generative
Shape Design Chapter 18:
Working with the
FreeStyle Workbench
Chapter 19: Introduction
to FEA and Generative
Structural Analysis Index
NX 12 Tutorial
Createspace Independent
Publishing Platform
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 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 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 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 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 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 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 of this book you will be ready to start printing out your own designs.

NX 2021 For Beginners
Createspace Independent Publishing Platform
The primary goal of Parametric Modeling with NX 12 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 NX 12 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 a hands-on, exercise-intensive

approach to all the important Parametric Modeling techniques and concepts. This textbook contains a series of fourteen tutorial style lessons designed to introduce beginning CAD users to NX. This text is also helpful to NX 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 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 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 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 of this book you will be ready to start printing out your own designs.

Practical Unigraphics SIAM Mold Design Using NX 11.0: A Tutorial Approach book is written with the intention of helping the

readers effectively design molds and its parts such as gate, runner, and various other standard parts using Mold Wizard of NX. After going through this book, the users will be able to design molds easily and effectively through processes such as analysis and documentation which have been dealt in detail. Also, the chapters in this book are arranged in a pedagogical sequence that makes this book very effective in learning the features and capabilities of the software. Keeping

in mind the requirements of the users, the book at first introduces basic terms and analyses and gradually progresses to cover sequential method to create mold and documentation. Written with the tutorial point of view and the learn by doing a theme, the book caters to the needs of both novice and advanced users and is ideally suited for learning at your convenience and pace. Salient Features Consists of 10 chapters that are organized in a pedagogical sequence.

Cover mold design concepts using NX 11.0. Tutorial approach to explain the concepts of Mold Design using NX 11.0. Summarized content on the first page of the topics that are covered in the chapter. Hundreds of illustrations for easy understanding of concepts. Step-by-step instructions to guide the users through the learning process. Additional information throughout the book in the form of notes and tips. Self-Evaluation Tests and Review Questions at the

end of each chapter to help the users assess their knowledge. Technical support by contacting techsupport@cadcim.com ' Additional learning resources at allaboutcadcam.blogspot.com' Table of Contents Chapter 1: Introduction to Mold Design and NX Mold Wizard Chapter 2: Part Analysis Chapter 3: Creating Parting Surface Chapter 4: Creating Core and Cavity Chapter 5: Adding Mold Base and Standard Parts Chapter 6: Creating Gate, Runner,

and Layout Chapter 7: Creating Sliders and Lifters Chapter 8: Creating Ejection and Cooling Systems Chapter 9: Creating Electrodes Chapter 10: Documentation Index **NX 2022 Tutorial** Createspace Independent Publishing Platform NX 2022 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 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, PMI, and rendering. Download Resource files from: online.books999@gmail.com *Containers in Cisco IOS-XE, IOS-XR, and NX-OS*

SDC Publications
The Primary Goal of Parametric Modeling with UGS NX 5 is to introduce the aspects of designing with Solid Modeling and Parametric Modeling. This text uses UGS NX as the modeling tool and the chapters guide you from constructing basic solid models to building intelligent mechanical designs, creating multi-view drawings and assembly models. This text takes a hands-on, exercise-intensive approach to all the important Parametric

Modeling techniques and concepts. This book contains a series of ten tutorial style lessons designed to introduce beginning CAD users to UGS NX. Each lesson introduces a new set of commands and concepts, building on previous lessons.

Mold Design Using NX 11.0: A Tutorial Approach
CAD/CIM Technologies
NX 2022 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 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, PMI, and rendering. Download Resource files from: online.books999@gmail.c

om

NX 9 Tutorial Cisco
Press

This textbook explains how to perform computer aided analysis by using NX 10 Advanced Simulation with NX Nastran solver. It starts with analyzing a cantilevered beam and builds up the reader's understanding of the concepts and process of structural analysis. Each chapter contains a typical example of analysis and is followed by a quiz to summarize the topics. In addition to the tutorial in

each chapter, more commands and concepts are explained at the end of the chapter to help improve the reader's understanding. The method for concluding an analysis is presented at the end of the tutorial for typical cases. Topics covered in this textbook - Chapter 1 through 3: Introducing NX 10 and Basic Modeling Techniques. - Chapter 4: Cantilevered Beam - Chapter 5: Effect of Fillet - Chapter 6: Effect of Stiffener - Chapter 7: Subcase and Symmetry -

Chapter 8: Static Equilibrium and Singularity - Chapter 9: Using Coordinate System in Constraining - Chapter 10: Using 2D Elements - Chapter 11: Using 1D Elements - Chapter 12: Analysis of Truss Structure - Chapter 13: Connecting 2D Meshes - Chapter 14: Using 1D and 2D Meshes - Chapter 15: Using 1D and 3D Meshes - Chapter 16: Analyzing Alternator Bracket - Chapter 17: Contact Analysis - Chapter 18: Analyzing Bearing and Housing - Chapter 19:

Spot Welding and Bolt Connection - Chapter 20: Analysis of Press Fit - Chapter 21: Quality of Elements - Chapter 22: Buckling Analysis - Chapter 23: Modal Analysis - Chapter 24: Thermal Analysis - Chapter 25: Fatigue Analysis

NX 8.5 Tutorial Book

Createspace Independent Publishing Platform

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/nx-11-tutorial>

Parametric Modeling with UGS NX5 SDC Publications

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

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 website. Almost all files are 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

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

Nx 11 Tutorial CAD/CIM Technologies
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 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 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 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 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 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 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 of this book you will be ready to start printing out your own

designs.

A Problem Book in Real Analysis Tutorial Books
Siemens NX 2021 for Designers is a comprehensive book that introduces the users to feature-based 3D parametric solid modeling using the NX software. The book covers all major environments of NX with a thorough explanation of all tools, options, and their applications to create real-world products. More than 40 mechanical engineering industry examples and additional 35 exercises

given in the book ensure that the users properly understand the solid modeling design techniques used in the industry and are able to efficiently create parts, assemblies, drawing views with bill of materials as well as learn the editing techniques that are essential to make a successful design. In this edition, four industry-specific projects are also provided for free download to the users to practice the tools learned and enhance their skills.

The NASTRAN

Theoretical Manual

Createspace Independent Publishing Platform

The third edition of this highly acclaimed undergraduate textbook is suitable for teaching all the mathematics for an undergraduate course in any of the physical sciences. As well as lucid descriptions of all the topics and many worked examples, it contains over 800 exercises. New stand-alone chapters give a systematic account of the 'special functions' of physical science, cover an extended range of

practical applications of complex variables, and give an introduction to quantum operators. Further tabulations, of relevance in statistics and numerical integration, have been added. In this edition, half of the exercises are provided with hints and answers and, in a separate manual available to both students and their teachers, complete worked solutions. The remaining exercises have no hints, answers or worked solutions and can be used for unaided homework;

full solutions are available to instructors on a password-protected web site,
www.cambridge.org/9780521679718.

**The Probability
 Tutoring Book** SDC
 Publications

Welcome to "NX For Beginners"! Master NX with step-by-step instructions. 2D sketches to 3D models. Assemble, create pro drawings. Explore Sheet Metal, Surface Design, Realize Shape. Unleash creativity! NX journey begins!
Simulations with NX

Courier Corporation
 This textbook explains how to create freeform surface and modify them to create freeform face of a solid body using Siemens NX 12. NX is a three dimensional CAD/CAM/CAE software developed by Siemens PLM Software Inc., Germany. Users of NX 9, 10 and 11 can use this book with minor modifications. We provide files for exercises via our website. Most of all files are 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 understand basic modeling process with NX. He/She has to be able to create sketch and fully constrain it, create the extruded and revolved features, apply boolean operation between solid bodies and understand how to use part navigator and selection toolbar. This textbook is suitable for anyone interested in creating mechanical surface and applying for solid body using Siemens NX. Topics covered in this textbook- Chapter 1: Basic

components of Siemens NX 12, options and mouse operations.- Chapter 2: Introduction to surface modeling process of NX 12.- Chapter 3 and 4: Creating Ruled and Through Curves surface.- Chapter 5: Face analysis.- Chapter 6, 7, 8 and 9: Creating Through Curve Mesh, Swept, Studio Surface and Variational Sweep surface.- Chapter 10: Commands for creating curves.- Chapter 11: Other helpful commands for creating surface model. - Chapter 12: Modeling projects.-

Chapter 13: Modeling bumper surface of Audi Q5.
Shape Interrogation for Computer Aided Design and Manufacturing
Independently Published
Siemens NX 2019 for Designers is a comprehensive book that introduces the users to feature based 3D parametric solid modeling using the NX software. The book covers all major environments of NX with a thorough explanation of all tools, options, and their applications to create real-world

products. In this book, about 40 mechanical engineering industry examples are used as tutorials and an additional 35 as exercises to ensure that the users can relate their knowledge and understand the design techniques used in the industry to design a product. After reading the book, the user will be able to create parts, assemblies, drawing views with bill of materials, and learn the editing techniques that are essential to make a successful design. Also, in

this book, the author emphasizes on the solid modeling techniques that improve the productivity and efficiency of the user. Keeping in mind the requirements of the users, the book at first introduces sketching and part modeling in NX, and then gradually progresses to cover assembly, surfacing, and drafting. To make the users understand the concepts of Mold Design, a chapter on mold designing of the plastic components is available in the book. In addition, a new chapter

on basic concepts of GD&T has also been added in this book. Both these chapters are available for free download. Written with the tutorial point of view and the learn-by-doing theme, the book caters to the needs of both novice and advanced users of NX and is ideally suited for learning at your convenience and pace. Salient Features: Comprehensive coverage of NX concepts and techniques. Tutorial approach to explain the concepts and tools of NX.

Detailed explanation of all commands and tools. Hundreds of illustrations for easy understanding of concepts. Step-by-step instructions to guide the users through the learning process. More than 40 real-world mechanical engineering designs as tutorials, 35 as exercises, and projects with step-by-step explanation. Additional information throughout the book in the form of notes and tips. Self-Evaluation Tests and Review Questions at the end of each chapter to help the users assess

their knowledge. Table of Contents Chapter 1: Introduction to NX Chapter 2: Drawing Sketches for Solid Models Chapter 3: Adding Geometric and Dimensional Constraints to Sketches Chapter 4: Editing, Extruding, and Revolving Sketches Chapter 5: Working with Datum Planes, Coordinate Systems, and Datum Axes Chapter 6: Advanced Modeling Tools-I Chapter 7: Advanced Modeling Tools-II Chapter 8: Assembly Modeling-I Chapter 9: Assembly

Modeling-II Chapter 10: Surface Modeling Chapter 11: Advanced Surface Modeling Chapter 12: Generating, Editing, and Dimensioning the Drawing Views Chapter 13: Synchronous Modeling Chapter 14: Sheet Metal Design Chapter 15: Introduction to Injection Mold Design (For Free Download) Chapter 16: Concepts of Geometric Dimensioning and Tolerancing (For Free Download) Index [Siemens Nx 12 Design Fundamentals](#) Springer Science & Business Media

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 3 D modeling using Siemens NX. [publisher's note] *Nx 10 for Beginners* CreateSpace Accessible but rigorous, this outstanding text encompasses all of the topics covered by a typical course in elementary abstract algebra. Its easy-to-read treatment offers an intuitive approach,

featuring informal discussions followed by thematically arranged exercises. This second edition features additional exercises to improve student familiarity with applications. 1990 edition. *NX 11 for Beginners* Hanser Publications

- Designed specifically for beginners with no prior CAD experience
- Uses a hands-on, exercise-intensive, tutorial style approach
- Covers parametric modeling, 3D Modeling, sheet metal design, assembly modeling, multiview

drawings and more • Includes chapters introducing you to 3D printing, advanced assembly modeling and animation 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 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 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 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 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 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 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 of this book you will be ready to start printing out your own designs. *CATIA V5-6R2017 for Designers, 15th Edition* CADCIM Technologies Siemens NX 12.0 for Designers is a comprehensive book that introduces the users to feature based 3D parametric solid modeling using the NX 12.0 software. The book covers all major environments of NX with a thorough explanation of all tools, options, and their applications to create

real-world products. In this book, about 39 mechanical engineering industry examples are used as tutorials and an additional 34 as exercises to ensure that the users can relate their knowledge and understand the design techniques used in the industry to design a product. After reading the book, the user will be able to create parts, assemblies, drawing views with bill of materials, and learn the editing techniques that are essential to make a

successful design. Also, in this book, the author emphasizes on the solid modeling techniques that improve the productivity and efficiency of the user. Salient Features: Consists of 16 chapters that are organized in a pedagogical sequence. Comprehensive coverage of NX 12.0 concepts and techniques. Tutorial approach to explain the concepts of NX 12.0. Hundreds of illustrations for easy understanding of concepts. More than 39 real-world mechanical engineering designs as

tutorials, 34 as exercises, and projects with step-by-step explanation. Additional information throughout the book in the form of notes and tips. Self-Evaluation Tests and Review Questions at the end of each chapter to help the users assess their knowledge. Technical support by contacting 'techsupport@cadcam.com'. Additional learning resources at 'allaboutcadcam.blogspot.com'. Table of Contents Chapter 1: Introduction to NX 12.0 Chapter 2:

Drawing Sketches for Solid Models Chapter 3: Adding Geometric and Dimensional Constraints to Sketches Chapter 4: Editing, Extruding, and Revolving Sketches Chapter 5: Working with Datum Planes, Coordinates Systems, and Datum Axes Chapter 6:

Advanced Modeling Tools-I Chapter 7: Advanced Modeling Tools-II Chapter 8: Assembly Modeling-I Chapter 9: Assembly Modeling-II Chapter 10: Surface Modeling Chapter 11: Advanced Surface Modeling Chapter 12: Generating, Editing, and Dimensioning the Drawing

Views Chapter 13: Synchronous Modeling Chapter 14: Sheet Metal Design Chapter 15: Introduction to Injection Mold Design (For Free Download) Chapter 16: Concepts of Geometric Dimensioning and Tolerancing (For Free Download) Index