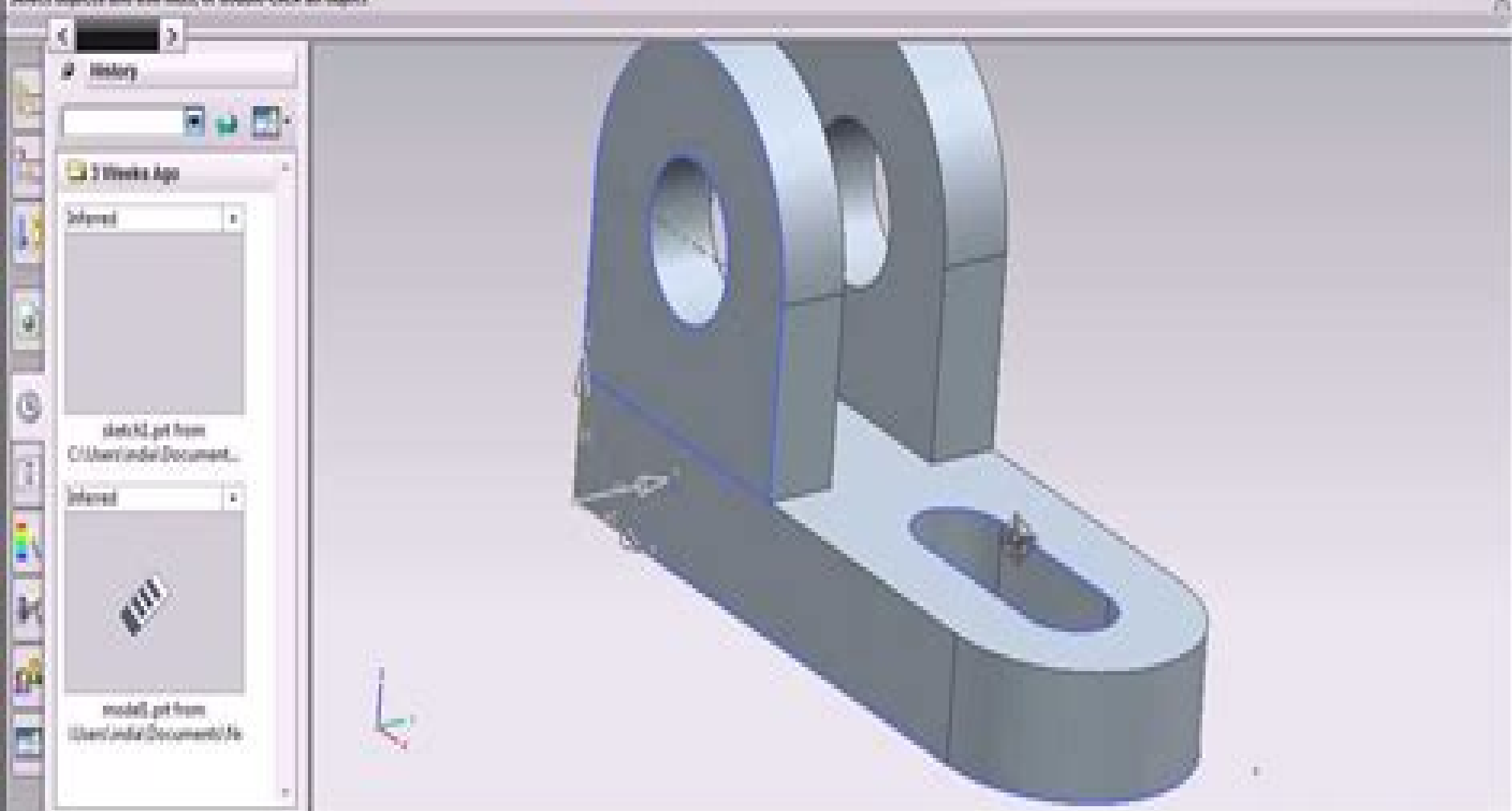


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Ug Nx 50 Advanced Design Tutorial Examples

Jaecheol Koh



Ug Nx 50 Advanced Design Tutorial Examples:

Siemens Nx 12 Design Fundamentals Jaecheol Koh, 2018-07-18 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](#) 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/nx11/tutorial> [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

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their knowledge and understand the design techniques used in the industry to design a product After reading the textbook 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 textbook at first introduces sketching and part modeling in NX 11 0 and then gradually progresses to cover assembly and drafting In addition a chapter on mold design for plastic components has been added in this textbook Written with the tutorial point of view and the learn by doing theme the textbook caters to the needs of both novice and advanced users of NX 11 0 and is ideally suited for learning at your convenience and pace

NX 10.0 for Designers Sham Tickoo, 2016 Consists of 800 pages of heavily illustrated text covering the following environments of NX part modeling assembly design surface design drafting synchronous modeling sheet metal design Cover

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