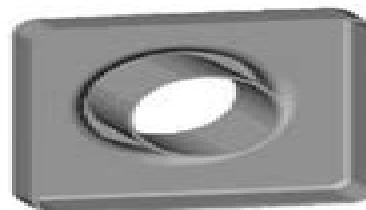


- Để có thêm thông tin và những tin mới nhất về công ty và phần mềm SolidWorks, hãy thăm web site, <http://www.solidworks.com> hoặc click Help, About SolidWorks 99, Connect.

CHƯƠNG 2

40 PHÚT KHỞI ĐẦU

Chương này sẽ hướng dẫn bạn tạo mô hình SolidWorks đầu tiên. Bạn sẽ tạo một chi tiết máy đơn giản như dưới đây:

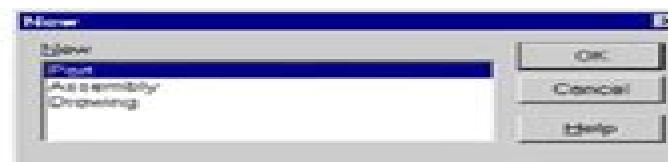


Chương này bao gồm:

- Tạo một hình khối cơ sở
- Thêm một phần lõi
- Khoét một lỗ thông
- Thay đổi các đặc điểm (thêm các góc lượn, thay đổi các kích thước)
- Hiển thị hình cắt của chi tiết máy
- Hiển thị nhiều công nhẫn của một chi tiết máy

Tạo một chi tiết máy mới

- Để tạo một chi tiết máy mới, Click New  trên thanh công cụ chuẩn, hoặc click File, New trên thanh menu. Hộp thoại New sẽ xuất hiện.
- Mỗi trường Part được chọn theo mặc định, click OK. Một cửa sổ Part mới sẽ xuất hiện.



Hiển thị các thanh công cụ

Các thanh công cụ cho phép ta nhanh chóng truy cập các tính năng và những lệnh thường dùng trong SolidWorks.

- Trên menu View, click Toolbar. Bạn phải nhìn thấy các thanh công cụ Standard, View, Features, Sketch và Standard View. Nếu muốn hiển thị thêm các thanh công cụ khác, ta có thể chọn chúng trên menu này. Dẫu sao, các thanh công cụ tương ứng cũng sẽ tự động hiển thị khi bạn mở các loại tài liệu khác nhau (Part, Assembly hay Drawing) hoặc mở một sketch.

[**Solidworks99 Tutorials Guide**](#)

Marie P. Planchard,David C. Planchard

Solidworks99 Tutorials Guide:

SolidWorks SolidWorks Corporation,1999 **SolidWorks 99** ,1999 *A Commands Guide Tutorial for SolidWorks*
2007 David C. Planchard,Marie P. Planchard,2007 **Introduction to SolidWorks** Godfrey C. Onwubolu,2017-03-03 This senior undergraduate level textbook is written for Advanced Manufacturing Additive Manufacturing as well as CAD CAM courses Its goal is to assist students in colleges and universities designers engineers and professionals interested in using SolidWorks as the design and 3D printing tool for emerging manufacturing technology for practical applications This textbook will bring a new dimension to SolidWorks by introducing readers to the role of SolidWorks in the relatively new manufacturing paradigm shift known as 3D Printing which is based on Additive Manufacturing AM technology This new textbook Features modeling of complex parts and surfaces Provides a step by step tutorial type approach with pictures showing how to model using SolidWorks Offers a user Friendly approach for the design of parts assemblies and drawings motion analysis and FEA topics Includes clarification of connections between SolidWorks and 3D Printing based on Additive Manufacturing Discusses a clear presentation of Additive Manufacturing for Designers using SolidWorks CAD software Introduction to SolidWorks A Comprehensive Guide with Applications in 3D Printing is written using a hands on approach which includes a significant number of pictorial descriptions of the steps that a student should follow to model parts assemble parts and produce drawings **SolidWorks 99** SolidWorks Corporation,1999 **SolidWorks 2007 Bible** Matt Lombard,2008-03-31 The most complete resource for SolidWorks on the market Matt Lombard s in depth knowledge plus his snappy wit and wisdom make SolidWorks accessible to users at all levels Mike Sabocheck Territory Technical Manager SolidWorks Corporation The most comprehensive single reference on SolidWorks Whether you re a new intermediate or professional user you ll find the in depth coverage you need to succeed with SolidWorks 2007 in this comprehensive reference From customizing the interface to exploring best practices to reinforcing your knowledge with step by step tutorials the techniques and shortcuts in this detailed book will help you accomplish tasks avoid the time consuming pitfalls of parametric design and get a firm handle on one of the leading 3D CAD programs on the market Customize the user interface and connect hotkeys to macros Create sketches parts assemblies and drawings Build intelligence into parts Work with patterns equations and configurations Learn multibody surface and master model techniques Write record and edit Visual Basic r macros Design with advanced 3D features Increase speed and efficiency with subassemblies Use multibody models to their full potential What s on the CD ROM The CD includes all the parts assemblies drawings and examples you need to follow the tutorials in each chapter You ll also find finished models templates and more See the CD appendix for details and complete system requirements **SolidWorks 2001 Tutorial** Marie P. Planchard,David C. Planchard,2001
Solidworks 2007 Tutorial with Multimedia CD David C. Planchard,Marie P. Planchard,2007 SolidWorks 2007 Tutorial was written to assist students designers engineers and professionals The book provides an introduction to the user

interface menus toolbars concepts and modeling techniques of SolidWorks to create parts assemblies and drawings Follow the step by step instructions and develop multiple assemblies that combine over 80 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables and configurations Review 2 hours of flash movie files that follow the steps in the book Desired outcomes and usage competencies are listed for each project Know your objective up front Follow the steps in Project 1 through Project 4 to achieve your design goals Work between multiple documents features commands and custom properties that represent how engineers and designers utilize SolidWorks in industry Table of Contents Introduction 1 Linkage Assembly 2 Front Support Assembly 3 Fundamentals of Drawing 4 Pneumatic Test Module Assembly Appendix Index **Drawing and Detailing with SolidWorks 2007** David C.

Planchard,Marie P. Planchard,2007 Drawing and Detailing with SolidWorks 2007 is written to educate and assist students designers engineers and professionals in the following areas A solid foundation using SolidWorks Drawing Options and SolidWorks Detailing Options Applying Engineering drawing standards and practices using SolidWorks tools Building multiple part and assembly configurations that interact with drawings Bill of Materials and Design Tables A comprehensive understanding of the differences between Drawing Templates and Sheet Formats Increase SolidWorks functionality to create view types with various configurations Combine a series of SolidWorks tools to solve a specific problem using Custom Properties and SolidWorks Properties The book utilizes a competency based approach on five projects Real world parts projects and tasks are addressed Commands are presented in a step by step progressive approach The learning process is explored through a series of design situations industry scenarios projects and objectives Table of Contents Introduction 1 Drawing Template and Sheet Format 2 Drawing View 3 Fundamentals of Detailing 4 Assembly Drawing 5 Applied Geometric Tolerancing and Other Symbols Appendix Index **Design Project Tutorial Using SolidWorks 99** Donald E. Coho,Schroff Development Corporation,1999-12 Mekanik Tasarım ve Analizin Bilimsel Esasları İlyas KACAR,2023-12-11 Heat Treating and Surface Engineering ASM Heat Treating Society. Conference and Exposition,2003-01-01 Mastering SolidWorks Matt Lombard,2018-10-26 The complete SolidWorks reference tutorial for beginner to advanced techniques

Mastering SolidWorks is the reference tutorial for all users Packed with step by step instructions video tutorials for over 40 chapters and coverage of little known techniques this book takes you from novice to power user with clear instruction that goes beyond the basics Fundamental techniques are detailed with real world examples for hands on learning and the companion website provides tutorial files for all exercises Even veteran users will find value in new techniques that make familiar tasks faster easier and more organized including advanced file management tools that simplify and streamline pre flight checks SolidWorks is the leading 3D CAD program and is an essential tool for engineers mechanical designers industrial designers and drafters around the world User friendly features such as drag and drop point and click and cut and

paste tools belie the software's powerful capabilities that can help you create cleaner more precise more polished designs in a fraction of the time. This book is the comprehensive reference every SolidWorks user needs with tutorials background and more for beginner to advanced techniques. Get a grasp on fundamental SolidWorks 2D and 3D tasks using realistic examples with text based tutorials. Delve into advanced functionality and capabilities not commonly covered by how to guides. Incorporate improved search Pack and Go and other file management tools into your workflow. Adopt best practices and exclusive techniques you won't find anywhere else. Work through this book beginning to end as a complete SolidWorks course or dip in as needed to learn new techniques and time saving tricks on demand. Organized for efficiency and designed for practicality these tips will remain useful at any stage of expertise. With exclusive coverage and informative detail Mastering SolidWorks is the tutorial reference for users at every level of expertise.

SOLIDWORKS 2024: A Power Guide for Beginners and Intermediate Users Sandeep Dogra, SOLIDWORKS 2024 A Power Guide for Beginners and Intermediate Users textbook is designed for both instructor led courses and self paced learning. It is intended to help engineers and designers interested in learning SOLIDWORKS for creating 3D mechanical designs. This is a valuable resource for new SOLIDWORKS users and a great teaching tool for classroom training. With 14 chapters and a total of 780 pages the content extensively covers key SOLIDWORKS environments such as Sketching, Part Modeling, Assembly and Drawing. This textbook teaches users to use SOLIDWORKS mechanical design software for creating parametric 3D solid components, assemblies and 2D drawings. Additionally a dedicated chapter is included to guide users in creating multiple configurations of a design. This textbook not only focuses on the usage of the tools and commands of SOLIDWORKS but also the concept of design. Every chapter in this textbook contains tutorials that provide users with step by step instructions for creating mechanical designs and drawings with ease. Additionally every chapter ends with practical hands on test drives that allow users to experience the user friendly and powerful technical capabilities of SOLIDWORKS.

Multi-body Dynamic Modeling of Multi-legged Robots Abhijit Mahapatra, Shibendu Shekhar Roy, Dilip Kumar Pratihar, 2020-02-27. This book describes the development of an integrated approach for generating the path and gait of realistic hexapod robotic systems. It discusses in detail locomotion with straight ahead, crab and turning motion capabilities in varying terrains like sloping surfaces, staircases and various user defined rough terrains. It also presents computer simulations and validation using Virtual Prototyping VP tools and real world experiments. The book also explores improving solutions by applying the developed nonlinear constrained inverse dynamics model of the system formulated as a coupled dynamical problem based on the Newton Euler NE approach and taking into account realistic environmental conditions. The approach is developed on the basis of rigid multi body modelling and the concept that there is no change in the configuration of the system in the short time span of collisions.

SolidWorks 2014 Reference Guide David Planchard, 2014. The SolidWorks 2014 Reference Guide is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2014. SolidWorks is an immense software package and no one book

can cover all topics for all users This book provides a centralized reference location to address many of the tools features and techniques of SolidWorks 2014 Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2014 software If you are completely new to SolidWorks you should read Chapter 1 in detail and complete Lesson 1 Lesson 2 and Lesson 3 in the SolidWorks Tutorials Videos are provided to introduce the new user to the basics of using SolidWorks 3D CAD software If you are familiar with an earlier release of SolidWorks you still might want to skim Chapter 1 to become acquainted with some of the commands menus and features that you have not used or you can simply jump to any section in any chapter Each chapter 18 total provides detailed PropertyManager information on key topics with individual standalone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature All models for the 240 plus tutorials are located on the enclosed book CD with their solution initial and final Learn by doing not just by reading Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables configurations and more The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2014 The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs The author developed the tutorials by combining his own industry experience with the knowledge of engineers department managers professors vendors and manufacturers He is directly involved with SolidWorks every day and his responsibilities go far beyond the creation of just a 3D model [Beginner's Guide to SOLIDWORKS 2021 - Level I](#) Alejandro Reyes,2021-01-29 This book is intended to help new users learn the basic concepts of SOLIDWORKS and good solid modeling techniques in an easy to follow guide that includes video instruction It is a great starting point for those new to SOLIDWORKS or as a teaching aid in classroom training to become familiar with the software s interface basic commands and strategies as users complete a series of models while learning different ways to accomplish a particular task At the end of this book you will have a fairly good understanding of the SOLIDWORKS interface and the most commonly used commands for part modeling assembly and detailing after completing a series of components and their 2D drawings complete with Bill of Materials The book focuses on the processes to complete the modeling of a part instead of focusing on individual software commands or operations which are generally simple enough to learn Throughout this book the author introduces you to new commands that are required to pass the Certified SOLIDWORKS Associate exam as listed on the SOLIDWORKS website A dedicated chapter provides you with details about the exam as well as a practice test to help you prepare for the actual exam SOLIDWORKS is an easy to use CAD software that includes many time saving tools that will enable new and experienced users to complete design tasks faster than before Most commands covered in this book have advanced options which may not be covered in this book This is meant to be a starting point to help new users to learn the basic and most frequently used commands

43rd AIAA Aerospace Sciences Meeting & Exhibit ,2005 Forthcoming Books Rose Arny,2002 **Commands Guide Tutorial**

for SolidWorks 2012 David C. Planchard, Marie P. Planchard, 2011-12-18 The Commands Guide Tutorial for SolidWorks 2012 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2012. SolidWorks is an immense software package and no one book can cover all topics for all users. The book provides a centralized reference location to address many of the tools, features, and techniques of SolidWorks 2012. This book covers the following: System and Document properties, FeatureManagers, PropertyManagers, ConfigurationManagers, RenderManagers, 2D and 3D Sketch tools, Sketch entities, 3D Feature tools, Motion Study, Sheet Metal, Motion Study, Sustainability, Sustainability, Xpress, FlowXpress, PhotoView 360, Pack and Go, Intelligent Modeling, techniques, and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2012 software. If you are completely new to SolidWorks, you should read Chapter 1 in detail and complete Lesson 1, Lesson 2, and Lesson 3 in the SolidWorks Tutorials. If you are familiar with an earlier release of SolidWorks, you still might want to skim Chapter 1 to become acquainted with some of the commands, menus, and features that you have not used or you can simply jump to any section in any chapter. Each chapter (18 total) provides detail PropertyManager information on key topics with individual stand alone short tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 240 plus tutorials are located on the enclosed book CD with their solution initial and final. Learn by doing, not just by reading. Formulate the skills to create, modify, and edit sketches and solid features. Learn the techniques to reuse features, parts, and assemblies through symmetry, patterns, copied components, design tables, configurations, and more. The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2012. The goal is to illustrate how multiple design situations and systematic steps combine to produce successful designs. The authors developed the tutorials by combining their own industry experience with the knowledge of engineers, department managers, professors, vendors, and manufacturers. These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model.

The book delves into Solidworks99 Tutorials Guide. Solidworks99 Tutorials Guide is a crucial topic that must be grasped by everyone, from students and scholars to the general public. This book will furnish comprehensive and in-depth insights into Solidworks99 Tutorials Guide, encompassing both the fundamentals and more intricate discussions.

1. This book is structured into several chapters, namely:

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- Chapter 2: Essential Elements of Solidworks99 Tutorials Guide
- Chapter 3: Solidworks99 Tutorials Guide in Everyday Life
- Chapter 4: Solidworks99 Tutorials Guide in Specific Contexts
- Chapter 5: Conclusion

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6. In chapter 5, the author will draw a conclusion about Solidworks99 Tutorials Guide. The final chapter will summarize the key points that have been discussed throughout the book.

This book is crafted in an easy-to-understand language and is complemented by engaging illustrations. It is highly recommended for anyone seeking to gain a comprehensive understanding of Solidworks99 Tutorials Guide.

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Table of Contents Solidworks99 Tutorials Guide

1. Understanding the eBook Solidworks99 Tutorials Guide
 - The Rise of Digital Reading Solidworks99 Tutorials Guide
 - Advantages of eBooks Over Traditional Books
2. Identifying Solidworks99 Tutorials Guide
 - Exploring Different Genres
 - Considering Fiction vs. Non-Fiction
 - Determining Your Reading Goals
3. Choosing the Right eBook Platform
 - Popular eBook Platforms
 - Features to Look for in an Solidworks99 Tutorials Guide
 - User-Friendly Interface
4. Exploring eBook Recommendations from Solidworks99 Tutorials Guide
 - Personalized Recommendations
 - Solidworks99 Tutorials Guide User Reviews and Ratings
 - Solidworks99 Tutorials Guide and Bestseller Lists
5. Accessing Solidworks99 Tutorials Guide Free and Paid eBooks
 - Solidworks99 Tutorials Guide Public Domain eBooks
 - Solidworks99 Tutorials Guide eBook Subscription Services
 - Solidworks99 Tutorials Guide Budget-Friendly Options
6. Navigating Solidworks99 Tutorials Guide eBook Formats
 - ePub, PDF, MOBI, and More
 - Solidworks99 Tutorials Guide Compatibility with Devices
 - Solidworks99 Tutorials Guide Enhanced eBook Features
7. Enhancing Your Reading Experience
 - Adjustable Fonts and Text Sizes of Solidworks99 Tutorials Guide
 - Highlighting and Note-Taking Solidworks99 Tutorials Guide
 - Interactive Elements Solidworks99 Tutorials Guide
8. Staying Engaged with Solidworks99 Tutorials Guide

- Joining Online Reading Communities
- Participating in Virtual Book Clubs
- Following Authors and Publishers Solidworks99 Tutorials Guide

9. Balancing eBooks and Physical Books Solidworks99 Tutorials Guide

- Benefits of a Digital Library
- Creating a Diverse Reading Collection Solidworks99 Tutorials Guide

10. Overcoming Reading Challenges

- Dealing with Digital Eye Strain
- Minimizing Distractions
- Managing Screen Time

11. Cultivating a Reading Routine Solidworks99 Tutorials Guide

- Setting Reading Goals Solidworks99 Tutorials Guide
- Carving Out Dedicated Reading Time

12. Sourcing Reliable Information of Solidworks99 Tutorials Guide

- Fact-Checking eBook Content of Solidworks99 Tutorials Guide
- Distinguishing Credible Sources

13. Promoting Lifelong Learning

- Utilizing eBooks for Skill Development
- Exploring Educational eBooks

14. Embracing eBook Trends

- Integration of Multimedia Elements
- Interactive and Gamified eBooks

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