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# Autodesk Inventor Files For A Manual Gearbox

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An Integrated Approach

Parametric Modeling with Autodesk Inventor 2020

Autodesk Authorized Publisher

Autodesk Inventor 2020: Introduction to Solid Modeling (Mixed Units) - Part 2

Autodesk Authorized Publisher: Working with Imported Geometry (Mixed Units)

Autodesk Inventor 2021: A Power Guide for Beginners and Intermediate Users

Parametric Modeling with Autodesk Inventor 2019

Autodesk Vault Professional 2019

Learning Autodesk Inventor 2018

Autodesk Authorized Publisher: Working with Imported Data

Autodesk Inventor 2018: Working with Imported Data

Autodesk Inventor 2021 and Engineering Graphics

Autodesk Inventor 2020: Introduction for Experienced 3D CAD Users (Mixed Units) - Part 2

Learning Autodesk Inventor 2013

Autodesk Authorized Publisher

Autodesk Inventor 2015 Essentials Plus

An Integrated Approach

Autodesk Inventor 2019

Autodesk Authorized Publisher

Learning Autodesk Inventor 2020

Working with Imported Geometry (Mixed Units): Autodesk Authorized Publisher

Autodesk Authorized Publisher

Mastering Autodesk Inventor 2013 and Autodesk Inventor LT 2013

Autodesk Inventor 2016 and Engineering Graphics

Learning Autodesk Inventor 2016

Working with Imported Geometry (Mixed Units) : Autodesk Authorized Publisher

Learning Autodesk Inventor 2010  
Modeling, Assembly and Analysis  
Autodesk Authorized Publisher  
Learning Autodesk Inventor 2022  
Autodesk Inventor 2021  
Autodesk Authorized Publisher  
Learning Autodesk Inventor 2021  
Mastering Autodesk Inventor 2010  
Parametric Modeling with Autodesk Inventor 2017  
Autodesk Inventor 2017  
Autodesk Inventor 2021: Advanced Assembly Modeling (Mixed Units): Autodesk Authorized Publisher

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## **YAZMIN MILLER**

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SDC Publications

This book will teach you everything you need to know to start using Autodesk Inventor 2021 with easy to understand, step-by-step tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of your robot design. An unassembled version of the same robot used throughout the book can be bundled with the book. No previous experience with Computer Aided Design(CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the Inventor interface and its basic tools. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-

view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of Inventor's powerful tools and commands that enable you to easily construct complex features in your models. Also included is coverage of gears, gear trains and spur gear creation using Autodesk Inventor. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanism, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the final chapter of this book you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D animations of your robot in action. There are many

books that show you how to perform individual tasks with Autodesk Inventor, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

*An Integrated Approach* John Wiley & Sons

This book will teach you everything you need to know to start using Autodesk Inventor 2022 with easy to understand, step-by-step tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of your robot design. An unassembled version of the same robot used throughout the book can be bundled with the book. No previous experience with Computer Aided Design(CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the Inventor interface and its basic tools. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of Inventor's powerful tools and commands that enable you to easily construct complex features in your models. Also included is coverage of gears, gear trains and spur gear creation using Autodesk Inventor. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic

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*Parametric Modeling with Autodesk Inventor 2020* Ascent, Center for Technical Knowledge

Autodesk(R) Vault Professional 2019: Data Management for Autodesk(R) Inventor(R) Users introduces the Autodesk Vault Professional 2019 software to Autodesk Inventor Users. This student guide is intended for Autodesk Inventor users who need to access their design files from the Autodesk Vault software. It provides an introduction to the Autodesk Vault Professional software and focuses on Autodesk Vault's features for managing design projects with the Autodesk Inventor software from a user's perspective. Students can use the Autodesk Vault Professional 2019 software and should use the Autodesk Inventor 2019 software to complete the exercises in this student guide. Note that this student guide does not cover administrative functionality. Hands-on exercises are included to reinforce how to manage the design workflow process using the Autodesk Vault

Professional software. Included with this student guide is a training Vault that can be used alongside a production Vault, to ensure that both Vaults can be accessed from the Autodesk Vault software. Topics Covered Introduction to Autodesk Vault Features Using the Autodesk Vault client Searching the Vault Working with non-CAD Files in the Vault Working with Inventor Files in the Vault Customizing the User Interface Data Management and Reusing Design Data Items and Bill of Materials Change Management Prerequisites Access to the 2019 version of the software. The practices and files included with this guide might not be compatible with prior versions. Students should have a good working knowledge of the Autodesk Inventor software.

Autodesk Authorized Publisher SDC Publications

Autodesk Inventor 2015 Essentials Plus provides the foundation for a hands-on course that covers basic and advanced Autodesk Inventor features used to create, edit, document, and print parts and assemblies. You learn about part and assembly modeling through real-world exercises. Autodesk Inventor 2015 Essentials Plus demonstrates critical CAD concepts, from basic sketching and modeling through advanced modeling techniques, as it equips you with the skills to master this powerful professional tool. The book walks you through every component of the software, including the user interface, toolbars, dialogue boxes, sketch tools, drawing views, assembly modeling, and more. Its unique modular organization puts key information at your fingertips, while step-by-step tutorials make it an ideal resource for self-learning. Packed with vivid illustrations and practical exercises that emphasize modern-day applications, Autodesk Inventor 2015 Essentials Plus will prepare you for work in the real

world. Each chapter is organized into four sections. Objectives, which describe the content and learning objectives; topic coverage, which presents a concise review of the topic; exercises, which present the workflow for a specific command or process through illustrated step-by-step instructions; and finally a checking your skills section, which tests your understanding of the material. Who Should Use This Manual? The manual is designed to be used in instructor-led courses, although you may also find it helpful as a self-paced learning tool. It is recommended that you have a working knowledge of Microsoft Windows as well as a working knowledge of mechanical design principles.

*Autodesk Inventor 2020: Introduction to Solid Modeling (Mixed Units) - Part 2* SDC Publications

Autodesk(R) Vault Professional 2020: Data Management for Autodesk(R) Inventor(R) Users introduces the Autodesk Vault Professional 2020 software to Autodesk Inventor Users. This guide is intended for Autodesk Inventor users who need to access their design files from the Autodesk Vault software. It provides an introduction to the Autodesk Vault Professional software and focuses on Autodesk Vault's features for managing design projects with the Autodesk Inventor software from a user's perspective. You can use the Autodesk Vault Professional 2020 software and should use the Autodesk Inventor 2020 software to complete the exercises in this guide. Note that this guide does not cover administrative functionality. Hands-on exercises are included to reinforce how to manage the design workflow process using the Autodesk Vault Professional software. Included with this guide is a training Vault that can be used alongside a production

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**Autodesk Authorized Publisher: Working with Imported Geometry (Mixed Units)** SDC Publications

Autodesk Inventor 2021 and Engineering Graphics: An Integrated Approach will teach you the principles of engineering graphics while instructing you on how to use the powerful 3D modeling capabilities of Autodesk Inventor 2021. Using step-by-step tutorials, this text will teach you how to create and read engineering drawings while becoming proficient at using the most common features of Autodesk Inventor. By the end of the book you will be fully prepared to take and pass the Autodesk Inventor Certified User Exam. This text is intended to be used as a training guide for students and professionals. The chapters in this text proceed in a pedagogical fashion to guide you from constructing basic shapes to making complete sets of engineering drawings. This text takes a hands-on, exercise-intensive approach to all the important concepts of Engineering Graphics, as well as in-depth discussions of parametric feature-based CAD techniques. This textbook contains a series of fifteen

chapters, with detailed step-by-step tutorial style lessons, designed to introduce beginning CAD users to the graphic language used in all branches of technical industry. This book does not attempt to cover all of Autodesk Inventor 2021'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. Autodesk Inventor 2021 Certified User Examination The content of this book covers the performance tasks that have been identified by Autodesk as being included on the Autodesk Inventor 2021 Certified User examination. Special reference guides show students where the performance tasks are covered in the book.

Autodesk Inventor 2021: A Power Guide for Beginners and Intermediate Users Ascent, Center for Technical Knowledge Autodesk Inventor 2020 and Engineering Graphics: An Integrated Approach will teach you the principles of engineering graphics while instructing you on how to use the powerful 3D modeling capabilities of Autodesk Inventor 2020. Using step-by-step tutorials, this text will teach you how to create and read engineering drawings while becoming proficient at using the most common features of Autodesk Inventor. By the end of the book you will be fully prepared to take and pass the Autodesk Inventor Certified User Exam. This text is intended to be used as a training guide for students and professionals. The chapters in this text proceed in a pedagogical fashion to guide you from constructing basic shapes to making complete sets of engineering drawings. This text takes a hands-on, exercise-intensive approach to all the important concepts of Engineering

Graphics, as well as in-depth discussions of parametric feature-based CAD techniques. This textbook contains a series of fifteen chapters, with detailed step-by-step tutorial style lessons, designed to introduce beginning CAD users to the graphic language used in all branches of technical industry. This book does not attempt to cover all of Autodesk Inventor 2020'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. Autodesk Inventor 2020 Certified User Examination The content of this book covers the performance tasks that have been identified by Autodesk as being included on the Autodesk Inventor 2020 Certified User examination. Special reference guides show students where the performance tasks are covered in the book.

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#### **Autodesk Vault Professional 2019** SDC Publications

The complete, real-world reference and tutorial for mastering Autodesk Inventor 2013 This completely updated and revised edition includes new content requested by readers and coverage of all of Inventor's latest features. Mastering Autodesk Inventor 2013 and Inventor LT 2013 starts with a basic hands-on tour of

the 3D design workflow and concludes with coverage of Inventor's built in programming tools. In between you'll find exercises and productivity tips as well as information on all aspects of the Inventor tools in Inventor LT to Inventor Professional. This detailed guide helps you quickly become proficient with everything from 3D parametric modeling design concepts and working with large assemblies to Weldment design and the routed systems features. Written by an Autodesk Certified Instructor with extensive experience using and teaching Inventor, this book features techniques and tactics not documented elsewhere, making this an invaluable reference that you'll turn to again and again. Helps you master Autodesk Inventor 2013 and Inventor LT 2013 and the fundamentals of 3D design Reviews how to effectively configure and use Inventor project files Shows you how to build and edit robust part models using basic and advanced tools Explores the tools used for designing sheet metal parts and how to copy assemblies for design reuse Covers large assembly strategies and reviews the ever-changing computer hardware landscape Other topics include conducting dynamic simulation and stress analysis, and working with Plastics design features and Inventor tooling for mold design

#### **Learning Autodesk Inventor 2018** CADArtifex

Most schools using Autodesk software first introduce students to the 2D features of AutoCAD and then go on to its 3D Capabilities. Inventor is usually reserved for the second or third course or for a solid modeling course. However, another possibility is to introduce students first to solid modeling using Inventor and then to introduce AutoCAD as a 2D product. Students learn to create

solid models using Inventor and then learn how to create working drawings of their 3D models using AutoCAD. This approach provides students with a strong understanding of the process used to create models and drawing in the industry. This book contains a series of tutorial style lessons designed to introduce Autodesk Inventor, AutoCAD, solid modeling, and parametric modeling. It uses a hands-on, exercise-intensive approach to all the import parametric modeling techniques and concepts. The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, creating multi-view drawings and assembly models. Introduction to Inventor 2012 and AutoCAD 2012 consists of ten chapters from Parametric Modeling using Inventor 2012 and six chapters from AutoCAD 2012 Tutorial-First Level: 2D Fundamentals. This book is used by Ohio State in their freshman engineering program.

#### **Autodesk Authorized Publisher: Working with Imported Data** SDC Publications

The Autodesk(R) Inventor(R) 2017: Working with Imported Geometry student guide teaches you how to work with data from other CAD platforms using the Autodesk Inventor software. Using this student guide, you will learn the various methods for importing data into Autodesk Inventor and how you can edit both imported solid and surface data. Additionally, you will learn how to index scanned point cloud data, and attach and use it in an Inventor file. The final chapters in this student guide discuss how you can use AutoCAD .DWG files in the Autodesk Inventor software. The topics covered in this student guide are also covered in ASCENT's Autodesk (R) Inventor (R) 2017: Advanced Part Modeling student guide, which includes a broader range of

advanced learning topics. Topics Covered Import CAD data into the Autodesk Inventor software. Export CAD data from the Autodesk Inventor software in an available export format. Index a supported point cloud data file, attach, and edit it for use in a file. Use the Edit Base Solid environment to edit solids that have been imported into the Autodesk Inventor software. Create Direct Edit features in a model that move, resize, scale, rotate, and delete existing geometry in both imported and native Autodesk Inventor files. Set the import options to import surface data from other file format types. Transfer imported surface data into the Repair Environment to conduct a quality check for errors. Appropriately set the stitch tolerance value so that gaps in the imported geometry can be automatically stitched and identify the gaps that are not stitched. Use the Repair Environment commands to repair gaps or delete, extend, replace, trim and break surfaces to successfully create a solid from the imported geometry. Open an AutoCAD DWG file directly into an Autodesk Inventor part file and review the data. Use the DWG/DXF File Wizard and its options to import files into an Autodesk Inventor file. Use an AutoCAD DWG file in an Autodesk Inventor part file so that the geometry created in Inventor remains associative with the AutoCAD DWG file. Freeform modeling. Emboss and Decal features. Advanced Drawing tools (iPart tables, surfaces in drawing views, and custom sketched symbols). Adding notes with the Engineer's Notebook. Prerequisites The material covered in this student guide assumes a mastery of Autodesk(R) Inventor(R) basics as taught in the Autodesk(R) Inventor(R) Introduction to Solid Modeling student guide.

**Autodesk Inventor 2018: Working with Imported Data** SDC

## Publications

Autodesk(R) Vault Professional 2021: Data Management for Autodesk(R) Inventor(R) Users introduces the Autodesk Vault Professional 2021 software to Autodesk Inventor users. This guide is intended for Autodesk Inventor users who need to access their design files from the Autodesk Vault software. It provides an introduction to the Autodesk Vault Professional software and focuses on Autodesk Vault's features for managing design projects with the Autodesk Inventor software from a user's perspective. You can use the Autodesk Vault Professional 2021 software and should use the Autodesk Inventor 2021 software to complete the exercises in this guide. Note that this guide does not cover administrative functionality. Hands-on exercises are included to reinforce how to manage the design workflow process using the Autodesk Vault Professional software. Included with this guide is a training Vault that can be used alongside a production Vault, to ensure that both Vaults can be accessed from the Autodesk Vault software. Topics Covered Introduction to Autodesk Vault Features Using the Autodesk Vault client Searching the Vault Working with non-CAD files in the Vault Working with Inventor files in the Vault Customizing the user interface Data management and reusing design data Items and bills of materials Change management Prerequisites Access to the 2021.0 version of the software, to ensure compatibility with this guide. Future software updates that are released by Autodesk may include changes that are not reflected in this guide. The practices and files included with this guide might not be compatible with prior versions (e.g., 2020). Good working knowledge of the Autodesk Inventor software.



### Autodesk Inventor 2021 and Engineering Graphics SDC Publications

Parametric Modeling with Autodesk Inventor 2019 contains a series of seventeen tutorial style lessons designed to introduce Autodesk Inventor, solid modeling, and parametric modeling. It uses a hands-on, exercise-intensive approach to all the important parametric modeling techniques and concepts. The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, to creating multi-view drawings and assembly models. Other featured topics include sheet metal design, motion analysis, 2D design reuse, collision and contact, stress analysis, 3D printing and the Autodesk Inventor 2019 Certified User Examination. Autodesk Inventor 2019 Certified User Examination The content of Parametric Modeling with Autodesk Inventor 2019 covers the performance tasks that have been identified by Autodesk as being included on the Autodesk Inventor 2019 Certified User examination. Special reference guides show students where the performance tasks are covered in the book. If you are teaching an introductory level Autodesk Inventor course and you want to prepare your students for the Autodesk Inventor 2019 Certified User Examination this is the only book that you need. If your students are not interested in the Autodesk Inventor 2019 Certified User Exam they will still be studying the most important tools and techniques of Autodesk Inventor as identified by Autodesk.

### Autodesk Inventor 2020: Introduction for Experienced 3D CAD Users (Mixed Units) - Part 2 John Wiley & Sons

The Autodesk® Inventor® 2018: Working with Imported Geometry student guide teaches you how to work with data from

other CAD platforms using the Autodesk Inventor software. Using this student guide, you will learn the various methods for importing data into Autodesk Inventor and how you can edit both imported solid and surface data. Additionally, you will learn how to index scanned point cloud data, and attach and use it in an Inventor file. The final chapters in this student guide discuss how you can use AutoCAD .DWG files in the Autodesk Inventor software. The topics covered in this student guide are also covered in ASCENT's Autodesk® Inventor® 2018: Advanced Part Modeling student guide, which includes a broader range of advanced learning topics. Topics covered: - Import CAD data into the Autodesk Inventor software. - Export CAD data from the Autodesk Inventor software in an available export format. - Index a supported point cloud data file, attach, and edit it for use in a file. - Use the Edit Base Solid environment to edit solids that have been imported into the Autodesk Inventor software. - Create Direct Edit features in a model that move, resize, scale, rotate, and delete existing geometry in both imported and native Autodesk Inventor files. - Set the import options to import surface data from other file format types. - Transfer imported surface data into the Repair Environment to conduct a quality check for errors. - Appropriately set the stitch tolerance value so that gaps in the imported geometry can be automatically stitched and identify the gaps that are not stitched. - Use the Repair Environment commands to repair gaps or delete, extend, replace, trim and break surfaces to successfully create a solid from the imported geometry. - Open an AutoCAD DWG file directly into an Autodesk Inventor part file and review the data. - Use the DWG/DXF File Wizard and its options to import files into an

Autodesk Inventor file. - Use an AutoCAD DWG file in an Autodesk Inventor part file so that the geometry created in Inventor remains associative with the AutoCAD DWG file. - Freeform modeling. - Emboss and Decal features. - Advanced Drawing tools (iPart tables, surfaces in drawing views, and custom sketched symbols). - Adding notes with the Engineer's Notebook.

Prerequisites: The material covered in this training guide assumes a mastery of Autodesk Inventor basics as taught in Autodesk® Inventor®: Introduction to Solid Modeling.

### **Learning Autodesk Inventor 2013** SDC Publications

Parametric Modeling with Autodesk Inventor 2020 contains a series of seventeen tutorial style lessons designed to introduce Autodesk Inventor, solid modeling, and parametric modeling. It uses a hands-on, exercise-intensive approach to all the important parametric modeling techniques and concepts. The lessons guide the user from constructing basic shapes to building intelligent mechanical designs, to creating multi-view drawings and assembly models. Other featured topics include sheet metal design, motion analysis, 2D design reuse, collision and contact, stress analysis, 3D printing and the Autodesk Inventor 2020 Certified User Examination. Autodesk Inventor 2020 Certified User Examination The content of Parametric Modeling with Autodesk Inventor 2020 covers the performance tasks that have been identified by Autodesk as being included on the Autodesk Inventor 2020 Certified User examination. Special reference guides show students where the performance tasks are covered in the book.

Autodesk Authorized Publisher SDC Publications

Learn Autodesk Inventor 2010 in this full-color Official Training

Guide This Official Training Guide from Autodesk is the perfect resource for beginners or professionals seeking training or preparing for certification in Autodesk's Inventor 3D mechanical design software. With instruction provided by experts who helped create the software, the book thoroughly covers Inventor principles and fundamentals, including 3D parametric part and assembly design, digital prototyping, and the creation of production-ready drawings. In eye-popping full color, the book includes pages of screen shots, step-by-step instruction, and real-world examples that both instruct and inspire. Takes you under the hood of Inventor 2010, Autodesk's 3D mechanical design software; this book is an Autodesk Official Training Guide Offers Autodesk's own, proven Inventor techniques, workflows, and content tailored to those developing their skills as well as professionals preparing for Inventor certification Teaches 3D parametric part and assembly design, digital prototyping, annotation, dimensioning, and drawing standards Demonstrates best practices for grouping parts into assemblies-then editing, manipulating, and creating drawings Illustrates in full-color with real-world designs, examples, and screen shots Learn Autodesk Inventor 2010 and prepare for Inventor certification with this in-depth guide.

Autodesk Inventor 2015 Essentials Plus ASCENT - Center for Technical Knowledge

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[An Integrated Approach](#) John Wiley & Sons

Note: This book is continued in Autodesk(R) Inventor(R) 2020: Introduction for Experienced 3D CAD Users - Part 2. Both books are required to complete this guide. The Autodesk(R) Inventor(R) 2020: Introduction for Experienced 3D CAD Users learning guide is intended to provide accelerated introductory training in the

Autodesk(R) Inventor(R) software. This learning guide is designed for users that have 3D modeling design experience with other 3D CAD software packages (e.g., CATIA(TM), Pro/ENGINEER(R), Creo Parametric(TM), NX(TM), SolidWorks(R), etc.). By leveraging the experience users gain in working with other 3D modeling software packages, this hands-on, practice-intensive guide is developed so that new users in the Autodesk Inventor software can benefit from a shorter, introductory-level, learning guide. You are taught how to find and use the modeling tools associated with familiar modeling strategies that are used in other 3D CAD software. You will acquire the knowledge required to complete the process of creating models from conceptual sketching, through to solid modeling, assembly design, and drawing production. Topics Covered The Autodesk Inventor software interface Obtaining model information Creating sketch and pick and place features Work Features Creating equations and working with parameters Model geometry and model display manipulation Feature duplication techniques Placing and constraining parts in assemblies Assembly component display Presentation files (Exploded views and Animations) Assembly tools Creating parts and features in assemblies Creating and editing assembly Bill of Materials Working with projects Creating and annotating drawings and views Prerequisites Access to the 2020.0 version of the software, to ensure compatibility with this guide. Future software updates that are released by Autodesk may include changes that are not reflected in this guide. The practices and files included with this guide are not compatible with prior versions (i.e., 2019). Prior knowledge of 3D modeling and 3D CAD software. Users with AutoCAD(R) or AutoCAD(R)

Mechanical experience are recommended to use the Autodesk Inventor 2020: Introduction to Solid Modeling guide.

**Autodesk Inventor 2019** SDC Publications

This book will teach you everything you need to know to start using Autodesk Inventor 2020 with easy to understand, step-by-step tutorials. This book features a simple robot design used as a project throughout the book. You will learn to model parts, create assemblies, run simulations and even create animations of your robot design. An unassembled version of the same robot used throughout the book can be bundled with the book. No previous experience with Computer Aided Design(CAD) is needed since this book starts at an introductory level. The author begins by getting you familiar with the Inventor interface and its basic tools. You will start by learning to model simple robot parts and before long you will graduate to creating more complex parts and multi-view drawings. Along the way you will learn the fundamentals of parametric modeling through the use of geometric constraints and relationships. You will also become familiar with many of Inventor's powerful tools and commands that enable you to easily construct complex features in your models. Also included is coverage of gears, gear trains and spur gear creation using Autodesk Inventor. This book continues by examining the different mechanisms commonly used in walking robots. You will learn the basic types of planar four-bar linkages commonly used in mechanical designs and how to use the GeoGebra Dynamic Geometry software to simulate and analyze 2D linkages. Using the knowledge you gained about linkages and mechanism, you will learn how to modify your robot and change its behavior by modifying or creating new parts. In the final chapter of this book

you learn how to combine all the robot parts into assemblies and then run motion analysis. You will finish off your project by creating 3D animations of your robot in action. There are many books that show you how to perform individual tasks with Autodesk Inventor, but this book takes you through an entire project and shows you the complete engineering process. By the end of this book you will have modeled and assembled nearly all the parts that make up the TAMIYA® Mechanical Tiger and can start building your own robot.

**Autodesk Authorized Publisher** SDC Publications

The Autodesk(R) Inventor(R) 2021: Advanced Assembly Modeling guide builds on the skills acquired in the Autodesk Inventor 2021: Introduction to Solid Modeling and Autodesk Inventor 2021: Advanced Part Modeling guides to take you to a higher level of productivity when creating and working with assemblies. You begin by focusing on the Top-Down Design workflow. You learn how tools are used to achieve this workflow using Derive, Multi-Body Design, and Layouts. Other topics include model simplification tools, Positional and Level of Detail Representations, iMates and iAssemblies, Frame Generator, Design Accelerator, and file management and duplication techniques. A chapter has also been included about the Autodesk(R) Inventor(R) Studio to teach you how to render, produce, and animate realistic images. Topics Covered Applying motion to existing assembly constraints using Motion and Transitional Constraints. Introduction of the Top-Down Design technique for creating assemblies and its components. Tools for Top-Down Design, such as associative links, adaptive parts, multi-body and layout design, derived components, and skeleton

models. Creating Positional Representations to review motion, evaluate the position of assembly components, or document an assembly in a drawing. Using Shrinkwrap and other model simplification tools to create a part model that represents an overall assembly. Creating Level of Detail Representations to reduce the clutter of large assemblies, reduce retrieval times, and substituting models. Using the Design Accelerator to easily insert standard and customizable components and features into your model. Creating rendered realistic images and animations of parts and assemblies using Autodesk Inventor Studio and the

Video Producer. Prerequisites Access to the 2021.0 version of the software, to ensure compatibility with this guide. Future software updates that are released by Autodesk may include changes that are not reflected in this guide. The practices and files included with this guide are not compatible with prior versions (e.g., 2020). The class assumes mastery of Autodesk Inventor basics as taught in Autodesk(R) Inventor(R) Introduction to Solid Modeling. In addition, Autodesk(R) Inventor(R) Advanced Part Modeling knowledge is recommended. The use of Microsoft(R) Excel is required for this training course.

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