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Solidworks  
Tutorial A Step By  
*Solidworks*  
Guide  
*Tutorial A*  
*Step By Guide*

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

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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 the user completes a series of

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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

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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

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are generally simple enough to learn. The author strived hard to include the commands required in the Certified SolidWorks Associate test as listed on the SolidWorks website, as well as several more. SolidWorks is an easy to use CAD

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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.

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## Tutorial A Step By Guide

This is meant to be a starting point to help new users to learn the basic and most frequently used commands.

SOLIDWORKS

2017 Advanced

Techniques picks up where

SOLIDWORKS

2017 Intermediate

Skills leaves off. Its

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aim is to take you from an intermediate user with a basic understanding of SOLIDWORKS and modeling techniques to an advanced user capable of creating complex models and able to use the advanced tools provided by SOLIDWORKS. The



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text covers parts, surfaces, SimulationXpress, sheet metal, top-down assemblies and core and cavity molds. Every lesson and exercise in this book was created based on real world projects. Each of these projects have been broken down

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and developed into easy and comprehensible steps. Furthermore, at the end of every chapter there are self test questionnaires to ensure that you have gained sufficient knowledge from each section before moving on to more advanced lessons.

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This book takes the approach that in order to understand SOLIDWORKS, inside and out, you should create everything from the beginning and take it step by step.

The only continuous, step-by-step tutorial for SolidWorks

SolidWorks is a 3D

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CAD manufacturing software package that has been used to design everything from aerospace robotics to bicycles. This book teaches beginners to use SolidWorks through a step-by-step tutorial, letting you build, document, and present a project

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while you learn.

Tools and functionality are explained in the context of professional, real-world tasks and workflows. You will learn the essential functions and gain the skills to use the software at once.

SolidWorks is a

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popular design software for manufacturing, and this book introduces it in the context of actually creating an object Begins with an overview of SolidWorks conventions and the interface Explains how to create models and drawings, create

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a revolved part and  
subassembly, and  
model parts within a  
subassembly

Explores

modification

capabilities and

drawing and Bill of

Materials templates

Moves on to top-level

assembly models and

drawings, Toolbox

components and the

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Design Library,  
mates, export and  
printing capabilities,  
and creating  
renderings Includes  
a glossary, a  
foreword from the  
SolidWorks product  
manager, and  
downloadable  
tutorial files

SolidWorks 2010: No  
Experience Required



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quickly turns  
beginners into  
confident users of  
SolidWorks.

SolidWorks 2014  
Tutorial with video  
instruction is  
targeted towards a  
technical school, two  
year college, four  
year university or  
industry professional  
that is a beginner or

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intermediate CAD user. The text provides a student who is looking for a step-by-step project based approach to learning SolidWorks with video instruction, SolidWorks model files, and preparation for the Certified Associate -

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Mechanical Design  
(CSWA) exam. The  
book is divided into  
two sections.

Chapters 1 - 5  
explore the  
SolidWorks User  
Interface and  
CommandManager,  
Document and  
System properties,  
simple machine  
parts, simple and

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## Tutorial A Step By Guide

complex assemblies,  
proper design intent,  
design tables,  
configurations, multi-  
sheet, multi-view  
drawings, BOMs,  
Revision tables using  
basic and advanced  
features. Chapters 6  
- 9 prepare you for  
the Certified  
Associate -  
Mechanical Design

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(CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D CAD and engineering practices and principles. Follow the step-by-step instructions and develop multiple assemblies that

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combine over 100  
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

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components, apply proper design intent, design tables and configurations.

Learn by doing, not just by reading.

Desired outcomes and usage

competencies are listed for each

chapter. Know your objective up front.

Follow the steps in

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each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document properties that represent how engineers and designers utilize SolidWorks in industry.



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A Power Guide for  
Beginners and  
Intermediate Users

SOLIDWORKS

2019 Quick Start  
Commands Guide

Tutorial for

Solidworks 2010

SOLIDWORKS

2022 Tutorial

Machining

Simulation Using

SOLIDWORKS

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CAM 2020

A comprehensive resource packed with information for both beginners and advanced users SolidWorks is the leading 3D solid modeling software used in computer-aided design. It's powerful but not

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## Tutorial A Step By Guide

simple. This complete guide introduces beginners to the software but then goes far beyond, covering numerous details that advanced users have requested. Beginners will learn not only how the software works but why, while more

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experienced users will learn all about search criteria, Pack-and-Go, other file management concepts, and much more. A valuable companion website contains before and after real-world parts and assemblies along with many example

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files used in the text. Additionally, the text of the book is augmented by video tutorials with author voice-over which can be found on the website. SolidWorks is the leading 3D CAD program, and previous editions of this book have sold

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more than 33,000  
copies Covers

necessary

information to give  
beginners a solid

foundation in the

software, including

part and assembly

modeling and 2D

drawing techniques

Addresses a wide

range of advanced

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topics not treated in other books, including best practices, search criteria, Pack-and-Go, and other file management concepts Includes tutorials on both beginning and advanced topics, with videos; sample part,

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assembly, and  
drawing files; and  
before-and-after  
example files available  
on the companion  
website SolidWorks  
2013 Bible is the  
ultimate resource on  
SolidWorks 2013, the  
book beginners can  
start with and  
advanced users will



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want to keep close at hand.

The Commands  
Guide Tutorial for  
SolidWorks 2010 is a  
comprehensive  
reference book  
written to assist  
beginner to  
intermediate users of  
SolidWorks.  
SolidWorks is an

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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 System and Document properties,

FeatureManagers,  
PropertyManagers, C

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Solidworks  
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onfiguration Manager  
s and

Render Managers  
along with 2D and  
3D Sketch tools,  
Sketch entities, 3D  
Feature tools, Motion  
Study,  
SustainabilityXpress,  
DFMXpress,  
SimulationXpress,  
Sheet Metal,

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PhotoView 360 and more. Chapter 1 provides a basic overview of the concepts and terminology used throughout this book using SolidWorks 2010 software. If you are completely new to SolidWorks, you should read Chapter

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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,

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menus and features that you have not used; or you can simply jump to any section in any chapter. Each chapter (17 total) provides detailed PropertyManager information on key topics with individual stand alone short

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tutorials to reinforce and demonstrate the functionality and ease of the SolidWorks tool or feature. All models for the 230 plus tutorials are located on the enclosed CD with their solution (initial and final). Learn by doing, not just by

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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



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more. The book is designed to compliment the Online Tutorials and Online Help contained in SolidWorks 2010. The goal is to illustrate how multiple design situations and systematic steps

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combine to produce  
successful designs.

The authors  
developed the  
tutorials by  
combining their own  
industry experience  
with the knowledge  
of engineers,  
department  
managers, vendors  
and manufacturers.

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These professionals are directly involved with SolidWorks everyday. Their responsibilities go far beyond the creation of just a 3D model.

**SOLIDWORKS  
2021: A Step-By-Step  
Tutorial Guide for  
Beginners (Mixed  
Units) textbook is**

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intended to help students, designers, engineers, and professionals who are interested in learning SOLIDWORKS step-by-step for creating real world 3D mechanical designs. It is a great starting point for new users of SOLIDWORKS and

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for those moving from other CAD software. This textbook contains tutorials that provide users with step-by-step instructions for creating parametric 3D solid components, assemblies, and 2D drawings with ease.

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Every tutorial in this textbook is created based on real-world projects. This textbook consists of 11 chapters, a total of 428 pages covering major environments of SOLIDWORKS such as Part modeling environment,  
Assembly

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environment, and  
Drawing  
environment  
including  
configurations. Every  
chapter ends with  
exercises that allow  
users to experience  
for themselves the  
user friendly and  
powerful capacities of  
SOLIDWORKS,

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which help users to assess their knowledge.

The Commands Guide Tutorial for SolidWorks 2013 is a comprehensive reference book written to assist the beginner to intermediate user of SolidWorks 2013.



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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 2013.

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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

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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

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using SolidWorks  
2013 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

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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

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detailed

Property Manager  
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

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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

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and assemblies through symmetry, patterns, copied components, design tables, configurations and more. The book is design to compliment the Online Tutorials and Online Help contained in SolidWorks 2013.



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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

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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

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of just a 3D model.

SOLIDWORKS 2018

Tutorial with Video

Instruction

SolidWorks 2015

Tutorial with Video

Instruction

Solidworks 2013

Bible

Solidworks 2020

- Uses step-by-

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step, project based tutorials designed for beginning or intermediate users • Will prepare you for the Certified SOLIDWORKS Associate Exam • Includes a chapter introducing you

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Tutorial A Step By  
Guide

to 3D printing  
SOLIDWORKS 2020

Tutorial is  
written to  
assist  
students,  
designers,  
engineers and  
professionals  
who are new to  
SOLIDWORKS. The  
text provides a  
step-by-step,

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project based learning approach. It also contains information and examples on the five categories in the CSWA exam. The book is divided into four sections. Chapters 1 - 5 explore the

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SOLIDWORKS User  
Interface and  
CommandManager,  
Document and  
System  
properties,  
simple and  
complex parts  
and assemblies,  
proper design  
intent, design  
tables,  
configurations,

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multi-sheet,  
multi-view  
drawings, BOMs,  
and Revision  
tables using  
basic and  
advanced  
features. In  
chapter 6 you  
will create the  
final robot  
assembly. The  
physical



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components and  
corresponding  
Science,  
Technology,  
Engineering and  
Math (STEM)  
curriculum are  
available from  
Gears  
Educational  
Systems. All  
assemblies and  
components for

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the final robot  
assembly are  
provided.

Chapters 7 - 10  
prepare you for  
the Certified  
Associate -  
Mechanical  
Design (CSWA)  
exam. The  
certification  
indicates a  
foundation in

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and apprentice  
knowledge of 3D  
CAD and  
engineering  
practices and  
principles.

Chapter 11  
covers the  
benefits of  
additive  
manufacturing  
(3D printing),  
how it differs

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from  
subtractive  
manufacturing,  
and its  
features. You  
will also learn  
the terms and  
technology used  
in low cost 3D  
printers.  
Follow the step-  
by-step  
instructions

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and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify and edit sketches and solid features.

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Learn the techniques to reuse features, parts and assemblies through symmetry, patterns, copied components, apply proper design intent, design tables

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configurations.  
Learn by doing,  
not just by  
reading.

Desired  
outcomes and  
usage  
competencies  
are listed for  
each chapter.

Know your  
objective up

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front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features, commands, custom properties and document



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properties that represent how engineers and designers utilize

SOLIDWORKS in industry.

SOLIDWORKS  
2020: A Step-By-  
Step Tutorial  
Guide for  
Beginners  
(Mixed Units)

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textbook is intended to help students, designers, engineers, and professionals who are interested in learning SOLIDWORKS step-by-step for creating real world 3D

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mechanical designs. It is a great starting point for new users of SOLIDWORKS and for those moving from other CAD software. This textbook contains tutorials that

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provide users  
with step-by-  
step  
instructions  
for creating  
parametric 3D  
solid  
components,  
assemblies, and  
2D drawings  
with ease.  
Every tutorial  
in this

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textbook is created based on real-world projects. This textbook consists of 11 chapters, a total of 428 pages covering major environments of SOLIDWORKS such as Part

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modeling  
environment,  
Assembly  
environment,  
and Drawing  
environment  
including  
configurations.  
Every chapter  
ends with  
exercises that  
allow users to  
experience for

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themselves the  
user friendly  
and powerful  
capacities of  
SOLIDWORKS,  
which help  
users to assess  
their  
knowledge.

SOLIDWORKS 2019  
Quick Start  
introduces the  
new user to the

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basics of using  
SOLIDWORKS 3D  
CAD software in  
five easy  
lessons. This  
book is  
intended for  
the student or  
designer that  
needs to learn  
SOLIDWORKS  
quickly and  
effectively for



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senior  
capstone,  
machine design,  
kinematics,  
dynamics, and  
other  
engineering and  
technology  
projects that  
use SOLIDWORKS  
as a tool.

Engineers in  
industry are

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expected to  
have SOLIDWORKS  
skills for  
their company's  
next project.  
Students need  
to learn  
SOLIDWORKS  
without taking  
a formal CAD  
course. Based  
on years of  
teaching

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SOLIDWORKS to  
engineering  
students,  
SOLIDWORKS 2019  
Quick Start  
concentrates on  
the areas where  
the new user  
improves  
efficiency in  
the design  
modeling  
process. By

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learning the correct SOLIDWORKS skills and file management techniques, you gain the most knowledge in the shortest period of time. You develop a mini Stirling Engine and

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investigate the proper design intent and constraints.

The mini Stirling Engine is based on the external combustion, closed cycle engine of Scottish inventor Robert

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Stirling. In addition to 3D modeling, the engine can be used to teach and connect many engineering and physics principles. You begin with an overview of SOLIDWORKS and

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Interface (UI),  
its menus,  
toolbars and  
commands. With  
a quick pace,  
you learn the  
essentials of  
2D sketching,  
part and  
assembly  
creation,  
perform motion

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study, develop  
detailed part  
and assembly  
drawings and  
much more.

This book will  
teach you all  
the important  
concepts and  
steps used to  
conduct  
machining  
simulations



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SOLIDWORKS CAM.

SOLIDWORKS CAM

is a

parametric,  
feature-based

machining

simulation

software

offered as an

add-in to

SOLIDWORKS. It

integrates

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design and  
manufacturing  
in one  
application,  
connecting  
design and  
manufacturing  
teams through a  
common software  
tool that  
facilitates  
product design  
using 3D solid

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models. By carrying out machining simulation, the machining process can be defined and verified early in the product design stage. Some, if not all, of the less desirable

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design features of part manufacturing can be detected and addressed while the product design is still being finalized. In addition, machining-related problems can be detected and

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eliminated  
before mounting  
a stock on a  
CNC machine,  
and  
manufacturing  
cost can be  
estimated using  
the machining  
time estimated  
in the  
machining  
simulation.

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This book is intentionally kept simple. It's written to help you become familiar with the practical applications of conducting machining simulations in SOLIDWORKS CAM. This book

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provides you with the basic concepts and steps needed to use the software, as well as a discussion of the G-codes generated. After completing this book, you

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should have a clear understanding of how to use SOLIDWORKS CAM for machining simulations and should be able to apply this knowledge to carry out machining assignments on



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product  
designs. In  
order to  
provide you  
with a more  
comprehensive  
understanding  
of machining  
simulations,  
the book  
discusses NC  
(numerical

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control) part programming and verification, as well as introduces applications that involve bringing the G-code post processed by SOLIDWORKS CAM to a HAAS CNC mill and lathe

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to physically  
cut parts. This  
book points out  
important,  
practical  
factors when  
transitioning  
from virtual to  
physical  
machining.  
Since the  
machining  
capabilities

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offered in the  
2020 version of  
SOLIDWORKS CAM  
are somewhat  
limited, this  
book introduces  
third-party CAM  
modules that  
are seamlessly  
integrated into  
SOLIDWORKS,  
including  
CAMWorks,

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HSMWorks, and  
Mastercam for  
SOLIDWORKS.

This book  
covers basic  
concepts,  
frequently used  
commands and  
options  
required for  
you to advance  
from a novice  
to an

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Tutorial A Step By  
intermediate  
level  
Guide

SOLIDWORKS CAM  
user. Basic  
concepts and  
commands  
introduced  
include  
extracting  
machinable  
features (such  
as 2.5 axis  
features),

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selecting a machine and cutting tools, defining machining parameters (such as feed rate, spindle speed, depth of cut, and so on), generating and simulating toolpaths, and

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post processing  
CL data to  
output G-code  
for support of  
physical  
machining. The  
concepts and  
commands are  
introduced in a  
tutorial style  
presentation  
using simple  
but realistic



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examples. Both milling and turning operations are included. One of the unique features of this book is the incorporation of the CL data verification by reviewing the G-

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code generated from the toolpaths. This helps you understand how the G-code is generated by using the respective post processors, which is an important step and an

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excellent way  
to confirm that  
the toolpaths  
and G-code  
generated are  
accurate and  
useful.

No Experience  
Required  
a step-by-step  
project based  
approach  
utilizing 3D

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solid modeling

Beginner's

Guide to

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- Level I

SOLIDWORKS 2020

Learn by Doing

Solidworks 2021

Engineering

Design with

SolidWorks 2011

is written to

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assist students,  
designers,  
engineers and  
professionals.

The book  
provides a solid  
foundation in  
SolidWorks by  
utilizing projects  
with step-by-  
step instructions  
for the beginning

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to intermediate  
SolidWorks user.  
Explore the user  
interface, Comm  
andManager,  
menus, toolbars  
and modeling  
techniques to  
create parts,  
assemblies and  
drawings in an  
engineering

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environment.

Follow the step-by-step instructions and develop multiple parts and assemblies that combine machined, plastic and sheet metal components.  
Formulate the

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skills to create,  
modify and edit  
sketches and  
solid features.

Learn the  
techniques to  
reuse features,  
parts and  
assemblies  
through  
symmetry,  
patterns, copied



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components,  
design tables,  
Bills of  
Materials,  
Custom  
Properties and  
Configurations.  
Address various  
SolidWorks  
analysis tools: Si  
mulationXpress,  
Sustainability / S

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ustainabilityXpre  
ss and

DFMXpress and  
Intelligent  
Modeling  
techniques.

Learn by doing,  
not just by  
reading! Desired  
outcomes and  
usage  
competencies

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are listed for  
each project.

Know your  
objective up  
front. Follow the  
steps in Project  
1 - 8 to achieve  
the design goals.  
Work between  
multiple  
documents,  
features,

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connecting design  
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teams through a  
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a discussion of the  
G-codes generated.  
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this book, you  
should have a clear  
understanding of  
how to use  
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simulations and  
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users to  
complete design  
tasks faster  
than before.  
Most commands  
covered in this  
book have  
advanced

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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. Includes Video Instruction



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is written to assist  
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professionals who  
are new to

SOLIDWORKS. The  
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fundamentals of  
engineering  
graphics and  
dimensioning  
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standard), line type,  
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fasteners in general,  
general thread  
notes and the  
history of CAD  
leading to the  
development of  
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Chapters 4 - 9:  
Comprehend the  
SOLIDWORKS User  
Interface and

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CommandManager,  
Document and  
System properties,  
simple machine  
parts, simple and  
complex assemblies,  
proper design  
intent, design  
tables,  
configurations,  
multi-sheet, multi-  
view drawings,  
BOMs, and Revision



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Associate (CSWA)  
exam. Understand  
the curriculum and  
categories of the  
CSWA exam and the  
required model  
knowledge needed  
to successfully take  
the exam. Chapter  
11: Provide a basic  
understanding  
between Additive

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vs. Subtractive  
manufacturing.

Discuss Fused  
Filament  
Fabrication (FFF),  
STereoLithography  
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technology. Select  
suitable filament  
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printer terminology.

Knowledge of  
preparing, saving,  
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own industry  
experience with the  
knowledge of  
engineers,  
department  
managers, vendors  
and manufacturers.  
SOLIDWORKS 2018  
Tutorial with video  
instruction is  
written to assist  
students, designers,  
engineers and



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professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also contains information and examples on the five categories, to take and understand the

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Certified Associate -  
Mechanical Design  
(CSWA) exam. The  
book is divided into  
four sections.

Chapters 1 - 5  
explore the  
SOLIDWORKS User  
Interface and  
CommandManager,  
Document and  
System properties,  
simple and complex

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parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter 6 you will create the final robot

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assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational Systems. All assemblies and

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components for the final robot assembly are provided.

Chapters 7 - 10 prepare you for the Certified Associate - Mechanical Design (CSWA) exam. The certification indicates a foundation in and apprentice knowledge of 3D

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CAD and engineering practices and principles. Chapter 11 covers the benefits of additive manufacturing (3D printing), how it differs from subtractive manufacturing, and its features. You will also learn the terms

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and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components. Formulate the skills to create, modify

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and edit sketches  
and solid features.

Learn the  
techniques to reuse  
features, parts and  
assemblies through  
symmetry, patterns,  
copied components,  
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intent, design tables  
and configurations.  
Learn by doing, not  
just by reading.



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Desired outcomes and usage competencies are listed for each chapter. Know your objective up front. Follow the steps in each chapter to achieve your design goals. Work between multiple documents, features,

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techniques  
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demands used  
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geometry or with  
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formulation by  
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plate results to thick-  
plate results

Introduction to  
Static Analysis using  
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students, educators,  
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professionals with  
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understanding of  
the features of SW  
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applicable to static analysis (FEA/FEM). SOLIDWORKS 2019 Tutorial is written to assist students, designers, engineers and professionals who are new to SOLIDWORKS. The text provides a step-by-step, project based learning approach. It also

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contains

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examples on the  
five categories in  
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Chapters 1 - 5

explore the  
SOLIDWORKS User  
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System properties, simple and complex parts and assemblies, proper design intent, design tables, configurations, multi-sheet, multi-view drawings, BOMs, and Revision tables using basic and advanced features. In chapter



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6 you will create the final robot assembly. The physical components and corresponding Science, Technology, Engineering and Math (STEM) curriculum are available from Gears Educational

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Systems. All assemblies and components for the final robot assembly are provided.

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apprentice  
knowledge of 3D  
CAD and  
engineering  
practices and  
principles. Chapter  
11 covers the  
benefits of additive  
manufacturing (3D  
printing), how it  
differs from  
subtractive  
manufacturing, and

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its features. You will also learn the terms and technology used in low cost 3D printers. Follow the step-by-step instructions and develop multiple assemblies that combine over 100 extruded machined parts and components.

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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, apply proper design intent, design tables and configurations.

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documents,  
features,  
commands, custom  
properties and  
document  
properties that  
represent how  
engineers and  
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provides a step-  
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Document and  
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properties,  
simple and  
complex parts  
and assemblies,  
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intent, design  
tables,  
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multi-sheet,**

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**multi-view  
drawings,  
BOMs, and  
Revision tables  
using basic and  
advanced  
features. In  
chapter 6 you  
will create the  
final robot  
assembly. The  
physical  
components**

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**corresponding  
Science,  
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**curriculum are  
available from  
Gears  
Educational  
Systems. All  
assemblies and**

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**components for  
the final robot  
assembly are  
provided.**

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prepare you for  
the Certified  
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exam. The  
certification  
indicates a**

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**foundation in  
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knowledge of  
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engineering  
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**Chapter 11  
covers the  
benefits of  
additive  
manufacturing  
(3D printing),**

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**how it differs  
from  
subtractive  
manufacturing,  
and its features.  
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learn the terms  
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used in low cost  
3D printers.  
Follow the step-  
by-step  
instructions**



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**and develop  
multiple  
assemblies that  
combine over  
100 extruded  
machined parts  
and  
components.  
Formulate the  
skills to create,  
modify and edit  
sketches and  
solid features.**

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patterns, copied  
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**configurations.  
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are listed for  
each chapter.**

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objective up  
front. Follow**

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**the steps in  
each chapter to  
achieve your  
design goals.  
Work between  
multiple  
documents,  
features,  
commands,  
custom  
properties and  
document  
properties that**

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design  
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process. By  
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correct  
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skills and file  
management  
techniques, you  
gain the most  
knowledge in  
the shortest**

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**This book  
begins with an  
overview of  
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and the User  
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toolbars and  
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and investigate  
the proper  
design intent  
and constraints.  
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