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drawings:
Standard,
Isometric,
Auxiliary, Section,

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with a Bill of
Materials and a
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SOLIDWORKS 2021
- Detailing Mode
and Drawing
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Detailing Mode
Enhancements -
Introduced in
SOLIDWORKS
2020, Detailing
Mode for Drawings
has been enhanced
to support for
creation of Detail,
Break and Crop
views. Hole callouts
are now supported
in Detailing Mode,

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content and characteristics for existing notes can be now be modified, and existing dimensions can be fully edited including tolerance values and arrow type.

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- Detailing Mode

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to make minor
edits to drawings of
large assemblies or

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configurations, or
resource-intensive
views. Detailing
mode is available
for drawings saved
in SOLIDWORKS
2020 and later.
Creating
Dimensions and
Annotations

2020 SOLIDWORKS

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2018.25.0.0_jx,
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In drawings, you

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can access context toolbars and menus for centerlines, sketches, drawing views, and markups. Location of the Hatch Pattern File You can save the hatch pattern file, sldwks.ptn, in any searchable folder so it is not

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This is useful if you
customize the file.

2021 What's New
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Detailing and
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Detailing Mode for
Drawings has been
enhanced in
SOLIDWORKS 2021

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to support the creation of Detail, Break and Crop views. Hole callouts are now supported in Detailing Mode, content and characteristics for existing notes can be now be modified, and existing dimensions can be fully edited

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Drawing and

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Enhancements

What's New

Drawing Templates
and Sheet Formats

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PAGE 5-4. When
opening a
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Detached drawing,
SolidWorks checks
all sheets in the
drawing to be sure
that they are
synchronized with
the model. If not,
you are warned. To
save a standard
drawing to a
Detached Drawing,
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2014
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to create multiple-
view, multiple-
sheet, detailed
drawings, and
assembly drawings.
Develop Drawing
templates, Sheet
formats, and
Custom Properties.

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2014
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that incorporate
part configurations,
assembly
configurations, and
design tables with
equations.

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annotations in
parts, drawings,
assemblies,
Revision tables,
Bills of Materials
and more. Apply

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detailing
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thirty exercises.
The exercises test
your usage
competency as
well as explore
additional topics
with industry
examples.
Advanced
exercises require
the ability to create

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Properties.
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that incorporate
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assembly

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and drawing techniques and tools. The book provides information and examples in the following areas: □

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isometric
projection, multi-
view drawings,
dimensioning
practices, fasteners
in general,
tolerance and fit
and the history of
CAD leading to the
development of
SOLIDWORKS. □

Start a
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session and to

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understand the following interfaces: Menu bar toolbar, Menu bar menu, Drop-down menus, Context toolbars, Consolidated drop-down toolbars, System feedback icons, Confirmation Corner, Heads-up View toolbar, Document

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with a Bill of
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Feature Control
Frames, Datums,
Geometric
Tolerancing,

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Surface Finishes,
and Weld Symbols
using Model Based
Definitions (MBD),
DimXpert and
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knowledge needed
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The author

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developed the
industry scenarios
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by combining his
own industry
experience with
the knowledge of
engineers,
department
managers, vendors
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These professionals
are directly
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projects, and
objectives targeted
towards the
beginning to
intermediate

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that incorporate

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icons, Confirmation
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7 provides a bonus
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SolidWorks models.

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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

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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

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understanding of
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the SOLIDWORKS
interface and the
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used commands
for part modeling,
assembly and
detailing after
completing a series
of components and
their 2D drawings
complete with Bill
of Materials. The

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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
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author introduces

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help you prepare
for the actual
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and experienced
users to complete
design tasks faster
than before. Most
commands covered

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in this book have
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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.

Engineering Design
with SOLIDWORKS

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2014
2021 is written to
assist students,
designers,
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professionals. The
book provides a
solid foundation in
SOLIDWORKS by
utilizing projects
with step-by-step
instructions for the
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featuring
machined, plastic
and sheet metal
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projects. Project 1 -
Project 6: Explore
the SOLIDWORKS

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BOMs, and Revision
tables using basic

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the edit and reuse
of features, parts,
and assemblies
through symmetry,
patterns,
configurations,
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ContentCentral and
the SOLIDWORKS
Toolbox. Project 7:
Understand Top-

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modeling and
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along with the
ability to import
parts using the Top-
Down assembly
method. Convert a
solid part into a
Sheet Metal part

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Sheet Metal
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Understand a
general overview of
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Simulation and the

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(CSWSA-FEA)
exam. Apply design
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intelligent
modeling
techniques in a
sketch, feature,
part, plane,

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drawing. Project
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the differences
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and subtractive
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Understand 3D
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along with a
working knowledge
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saving, and
printing CAD

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models on a low
cost printer. Project
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knowledge needed
to successfully take

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the exam. The
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the industry
scenarios by
combining his own
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experience with
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managers, vendors
and manufacturers.
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are directly

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every day. Their
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responsibilities go
far beyond the
creation of just a
3D model.

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students,

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Construct drawings
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tools. The book
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sketching
techniques,
orthographic
projection,
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projection, multi-
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practices, fasteners
in general,
tolerance and fit

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Consolidated drop-

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icons, Confirmation
Corner, Heads-up
View toolbar,
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drawings from
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configurations and
develop the

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Projected Back,
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Materials and a
Revision Table and
Revisions. Insert
and edit:
Dimensions,

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Frames, Datums,
Geometric
Tolerancing,
Surface Finishes,
and Weld Symbols
using DimXpert
and manual
techniques. Create,
apply, and save
Blocks and
Parametric Notes in
a drawing. Chapter
10 provides a

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Associate CSWA
program with
sample exam
questions and
initial and final
SolidWorks models.
The book is
designed to
compliment the
SolidWorks Users
Guide, SolidWorks

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Standards,
Engineering
Drawing/Design
and Graphics
Communications
reference books.
The authors
recognize that
companies utilize
additional drawing
standards. The
authors developed
the industry

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Scenarios by
combining industry
SolidWorks
2014
experience with
their knowledge of
engineers, sales,
vendors and
manufacturers.
These professionals
are directly
involved with
SolidWorks
everyday. Their
work goes far
beyond a simple

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drawing with a few
dimensions. They
create detailed
drawings, assembly
drawings,
marketing
drawings and
customer drawings.
SolidWorks users
work between
drawings, parts,
assemblies and
many other
documents to

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complete a project
on time.

2014

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

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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

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a particular task. At
the end of this
Solidworks
2014
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

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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

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learn. The author
strived hard to
include the
commands
required in the
Certified
SOLIDWORKS
Associate and
Certified
SOLIDWORKS
Professional Exams
as listed on the
SOLIDWORKS
website.

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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

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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.

The intention of
this book is to
provide you with a
guide for designing
package structure

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and packaging with
mechanism with
SolidWorks
2012.
2014
The intended
audience is the
student, engineer
and persons
involved in
designing
packaging
structure, product
design and
mechanical design.
The design

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examples in the
chapters are
Solidworks
2014
simplified versions
of real world
designs. The
chapter sequences
are arranged from
basic skills to more
advanced skills. It
is assumed that
the reader has
some familiarity
with SolidWorks.
Topics Covered

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modeling The Helix
tool and Reference
Plane Trim and
mirror entities
Assembling many
part drawings into
a more
complicated
assemble drawing
3-point Arcs and
copying sketches
from one plane to
another plane

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cut drawing Cavity
and split
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designing a mold
Calculating fill
volumes and using
PhotoView 360
Creating a bill of
materials for
assembly
instructions and
documentation

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complicated
components
Solidworks
2014
Creating 2D detail
drawings from part
drawings Advanced
mate,
subassembly,
adding screws and
nuts based on the
Design Library
Creating the
exploded views

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