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Introduction
To Creo
Simulate 2
Inneo

***The eleven
lessons in this
tutorial
introduce you
to the design
capabilities of***

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Creo

Parametric

9.0. The

tutorial covers

the major

concepts and

frequently

used

commands

required to

advance from

a novice to an

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***intermediate
user level.***

***Major topics
include part
and assembly
creation, and
creation of
engineering
drawings. Also
illustrated are
the major
functions that***

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***make Creo
Parametric a
parametric
solid modeler.
Although the
commands are
presented in a
click-by-click
manner, an
effort has been
made, in
addition to sho***

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***wing/illustrati
ng the
command
usage, to
explain why
certain
commands are
being used
and the
relation of
feature
selection and***

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**construction
to the overall
part design
philosophy.**

**Simply
knowing where
commands can
be found is
only half the
battle. As is
pointed out
numerous**

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***times in the
text, creating
useful and
effective
models of
parts and
assemblies
requires
advance
planning and
forethought.
Moreover,***

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since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally

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***induced so
that users will
become
comfortable
with the
“debugging”
phase of
model
creation. At
the end of
each lesson is
a short quiz***

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***reviewing the
new topics
covered in that
chapter.***

***Following the
quiz are
several simple
"exercise"
parts that can
be created
using new
commands***

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***taught in that
lesson. In
addition to
these an
ongoing
project
throughout
the book is
also included.
This project
consists of
several parts***

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***that are
introduced
with the early
lessons and
finally
assembled at
the end. Who
this book is for
This book has
been written
specifically
with students***

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in mind.

***Typically,
students enter
their first CAD
course with a
broad range of
abilities both
in spatial
visualization
and computer
skills. The
approach***

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***taken here is
meant to allow
accessibility to
persons of all
levels. These
lessons,
therefore,
were written
for new users
with no
previous
experience***

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with CAD,
Inno
*although some
familiarity
with
computers is
assumed. The
tutorials in
this textbook
cover the
following
topics: •*

Introduction

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*to the program
and its
operation •
The features
used in part
creation •
Modeling
utilities •
Creating
engineering
drawings •
Creating*

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assemblies

*and assembly
drawings*

- *Written for
first time FEA
and Creo*

Simulate users

- *Uses simple
examples with
step-by-step
tutorials •*

Explains the

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***relation of
commands to
the overall
FEA
philosophy •
Both 2D and
3D problems
are covered
Creo Simulate
9.0 Tutorial
introduces
new users to***

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***finite element
analysis using
Creo Simulate
and how it can
be used to
analyze a
variety of
problems. The
tutorial
lessons cover
the major
concepts and***

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*frequently
used
commands
required to
progress from
a novice to an
intermediate
user level. The
commands are
presented in a
click-by-click
manner using*

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simple
examples and
exercises that
illustrate a
broad range of
the analysis
types that can
be performed.
In addition to
showing the
command
usage, the text

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**will explain
why certain
commands are
being used
and, where
appropriate,
the relation of
commands to
the overall
Finite Element
Analysis (FEA)
philosophy are**

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explained.

**Moreover,
since error
analysis is an
important
skill,
considerable
time is spent
exploring the
created
models so that
users will**

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***become
comfortable
with the
“debugging”
phase of
modeling. This
textbook is
written for
first-time FEA
users in
general and
Creo Simulate***

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**users in
particular.
After a brief
introduction
to finite
element
modeling, the
tutorial
introduces the
major
concepts
behind the use**

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Creo Simulate 2
**of Creo
Simulate to
perform Finite
Element
Analysis of
parts. These
include modes
of operation,
element types,
design studies
(analysis,
sensitivity**

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Introduction To
Creo Simulate 2
***studies,
organization),
and the major
steps for
setting up a
model
(materials,
loads,
constraints,
analysis type),
studying
convergence of***

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*the solution,
and viewing
the results.
Both 2D and
3D problems
are covered.
This tutorial
deals
exclusively
with operation
in integrated
mode with*

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Creo

Parametric. It is suitable for use with both Releases 9.0 of Creo Simulate. The tutorials consist of the following: • 2 lessons on general introductory

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**material • 2
lessons
introducing
the basic
operations in
Creo Simulate
using solid
models • 4
lessons on
model
idealizations
(shells, beams**

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Creo Simulate 2
Inneo
**and frames,
plane stress,
etc) • 1 lesson
on
miscellaneous
topics • 1
lesson on
steady and
transient
thermal
analysis Table
of Contents 1.**

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Introduction To

Creo Simulate 2

**Introduction
to FEA 2.**

**Finite Element
Analysis with
Creo Simulate
3. Solid**

**Models Part 1:
Standard**

Static Analysis

4. Solid

**Models Part 2:
Design**

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Studies,
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AutoGEM
Controls,
Superposition
5. Plane Stress
and Plane
Strain Models
6.
Axisymmetric
Solids and
Shells 7. Shell

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**Models 8.
Beams and
Frames 9.
Miscellaneous
Topics: Cyclic
Symmetry,
Modal
Analysis,
Springs and
Masses,
Contact
Analysis 10.**

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***Thermal
Models:
Steady state
and transient
models;
transferring
thermal
results for
stress analysis
Designing with
Creo
Parametric 6.0***

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***provides the
high school
student,
college
student, or
practicing
engineer with
a basic
introduction
to engineering
design while
learning the***

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***3D modeling C
omputer-Aided
Design***

***software called
Creo***

Parametric

***from PTC. The
topics are***

presented in

tutorial format

with exercises

at the end of

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***each chapter
to reinforce
the concepts
covered. It is
richly
illustrated
with computer
screen shots
throughout.
Above all, this
text is
designed to***

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**help you
expand your
creative
talents and
communicate
your ideas
through the
graphics
language.
Because it is
easier to learn
new**

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information if

you have a

reason for

learning it,

this textbook

discusses

design intent

while you are

learning Creo

Parametric. At

the same time,

it shows how

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***knowledge
covered in
basic
engineering
courses such
as statics,
dynamics,
strength of
materials, and
design of
mechanical
components***

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***can be applied
to design. You
do not need an
engineering
degree nor be
working
toward a
degree in
engineering to
use this
textbook.***

Although FEA

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***(Finite
Element
Analysis) is
used in this
textbook, its
theory is not
covered. The
first two
chapters of
this book
describe the
design***

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***process. The
meat of this
text, learning
the basic Creo
Parametric
software, is
found in
Chapters 3
through 6.
Chapters 7, 8,
and 12 deal
with***

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***dimensioning
and
tolerancing an
engineering
part. Chapters
9 and 10 deal
with
assemblies
and assembly
drawings.
Chapter 11
deals with***

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*family tables
used when
similar parts
are to be
designed or
used. Chapter
13 is an
introduction
to Creo
Simulate and
FEA.*

Creo Simulate

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Introduction To
Creo Simulate 2
Tutorial
Releases 1.0 &
2.0 introduces
new users to
finite element
analysis using
Creo Simulate
and how it can
be used to
analyze a
variety of
problems. The

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tutorial
lessons cover
the major
concepts and
frequently
used
commands
required to
progress from
a novice to an
intermediate
user level. The

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commands are

presented in a

click-by-click

manner using

simple

examples and

exercises that

illustrate a

broad range of

the analysis

types that can

be performed.

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***In addition to
showing the
command***

***usage, the text
will explain***

why certain

***commands are
being used***

and, where

appropriate,

***the relation of
commands to***

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***the overall
Finite Element
Analysis (FEA)
philosophy are
explained.
Moreover,
since error
analysis is an
important
skill,
considerable
time is spent***

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***exploring the
created
models so that
users will
become
comfortable
with the
“debugging”
phase of
modeling. This
textbook is
written for***

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***first-time FEA
users in
general and
Creo Simulate
users in
particular.
After a brief
introduction
to finite
element
modeling, the
tutorial***

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**introduces the
major
concepts
behind the use
of Creo
Simulate to
perform Finite
Element
Analysis of
parts. These
include: modes
of operation,**

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element types,

design studies

(analysis,

sensitivity

studies,

organization),

and the major

steps for

setting up a

model

(materials,

loads,

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**constraints,
analysis type),
studying**

**convergence of
the solution,
and viewing
the results.**

**Both 2D and
3D problems
are treated.**

**This tutorial
deals**

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exclusively
with operation
in integrated
mode with
Creo
Parametric. It
is suitable for
use with both
Releases 1.0
and 2.0 of
Creo Simulate.
The primary

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goal of
***Introduction
to Finite
Element
Analysis Using
Creo Simulate
7.0 is to
introduce the
aspects of
finite element
analysis (FEA)
that are***

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***important to
engineers and
designers.***

***Theoretical
aspects of
finite element
analysis are
also
introduced as
they are
needed to help
better***

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***understand
the operations.***

***The primary
emphasis of
the text is
placed on the
practical
concepts and
procedures of
using Creo
Simulate in
performing***

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***Linear Statics
Stress***

***Analysis; but
the basic
modal analysis
procedure is
covered. This
text is
intended to be
used as a
training guide
for both***

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***students and
professionals.
This text
covers Creo
Simulate 7.0
and the
lessons
proceed in a
pedagogical
fashion to
guide you from
constructing***

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***basic truss
elements to
generating thr
ee-
dimensional
solid elements
from solid
models. This
text takes a
hands-on
exercise
intensive***

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***approach to all
the important
Finite Element
Analysis
techniques
and concepts.
This textbook
contains a
series of
twelve tutorial
style lessons
designed to***

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***introduce
beginning FEA
users to Creo
Simulate. The
basic premise
of this book is
the more
designs you
create using
Creo Simulate,
the better you
learn the***

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***software. With
this in mind,
each lesson
introduces a
new set of
commands and
concepts,
building on
previous
lessons.
Introduction
to Finite***

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Element
Inpec
Analysis Using
Creo Simulate
8.0
Creo
Parametric 1.0
Creo
Parametric 5.0
Tutorial
Creo Simulate
3.0 Tutorial
Creo Simulate

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Creo Simulate 2
7.0 Tutorial
Inneo

Mechanism Design
and Analysis Using
PTC Creo

Mechanism 6.0 is
designed to help
you become
familiar with
Mechanism, a
module of the PTC
Creo Parametric
software family,
which supports

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modeling and
analysis (or
simulation) of
mechanisms in a
virtual (computer)
environment.
Capabilities in
Mechanism allow
users to simulate
and visualize
mechanism
performance. Using
Mechanism early in

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the product
development stage
could prevent
costly redesign due
to design defects
found in the
physical testing
phase; therefore, it
contributes to a
more cost effective,
reliable, and
efficient product
development

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process. The book is written following a project-based learning approach and covers the major concepts and frequently used commands required to advance readers from a novice to an intermediate level. Basic concepts

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discussed include

model creation,

such as body and

joint definitions;

analysis type

selection, such as

static (assembly)

analysis, kinematics

and dynamics; and

results

visualization. The

concepts are

introduced using

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simple, yet realistic,
examples. Verifying
the results obtained
from computer
simulation is
extremely
important. One of
the unique features
of this textbook is
the incorporation of
theoretical
discussions for
kinematic and

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dynamic analyses in conjunction with simulation results obtained using Mechanism. The theoretical discussions simply support the verification of simulation results rather than providing an in-depth discussion on

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the subjects of
kinematics and
dynamics.

Designing with
Creo Parametric 8.0
provides the high
school student,
college student, or
practicing engineer
with a basic
introduction to
engineering design
while learning the

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3D modeling
Computer-Aided
Design software
called Creo
Parametric from
PTC. The topics are
presented in
tutorial format with
exercises at the end
of each chapter to
reinforce the
concepts covered. It
is richly illustrated

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with computer
screen shots
throughout. Above
all, this text is
designed to help
you expand your
creative talents and
communicate your
ideas through the
graphics language.
Because it is easier
to learn new
information if you

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have a reason for learning it, this textbook discusses design intent while you are learning Creo Parametric. At the same time, it shows how knowledge covered in basic engineering courses such as statics, dynamics,

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strength of
materials, and
design of
mechanical
components can be
applied to design.
You do not need an
engineering degree
nor be working
toward a degree in
engineering to use
this textbook.

Although FEA

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(Finite Element
Analysis) is used in
this textbook, its
theory is not
covered. The first
two chapters of this
book describe the
design process. The
meat of this text,
learning the basic
Creo Parametric
software, is found
in Chapters three

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through six.

Chapters seven, eight, and 12 deal with dimensioning and tolerancing an engineering part. Chapters nine and ten deal with assemblies and assembly drawings. Chapter 11 deals with family tables used when similar

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parts are to be
designed or used.
Chapter 13 is an
introduction to
Creo Simulate and
FEA. Table of
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Design 2.
Introduction 3.
Sketcher 4.
Extrusions 5.
Revolves 6. Patterns

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7. Dimensioning 8.

Engineering

Drawings 9.

Assemblies 10.

Assembly Drawings

11. Relations and

Family Tables 12.

Tolerancing and

GD&T 13. Creo

Simulate and FEA

Appendix A:

Parameters for
Drawings Appendix

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B: Drill and Tap

Chart Appendix C:

Surface Roughness

Chart Appendix D:

Clevis Pin Sizes

Appendix E:

Number and Letter

Drill Sizes Appendix

F: Square and Flat

Key Sizes Appendix

G: Screw Sizes

Appendix H: Nut

Sizes Appendix I:

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Setscrew Sizes
Appendix J: Washer
Sizes Appendix K:
Retaining Ring Sizes
Appendix L: Basic
Hole Tolerance
Appendix M: Basic
Shaft Tolerance
Appendix N:
Tolerance Zones
Appendix O:
International
Tolerance Grades

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References Index

The primary goal of Introduction to Finite Element Analysis Using Creo Simulate 8.0 is to introduce the aspects of finite element analysis (FEA) that are important to engineers and designers.

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Theoretical aspects of finite element analysis are also introduced as they are needed to help better understand the operations. The primary emphasis of the text is placed on the practical concepts and procedures of using Creo Simulate in

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performing Linear
Statics Stress
Analysis; but the
basic modal
analysis procedure
is covered. This text
is intended to be
used as a training
guide for both
students and
professionals. This
text covers Creo
Simulate 8.0 and

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the lessons proceed

in a pedagogical

fashion to guide

you from

constructing basic

truss elements to

generating three-

dimensional solid

elements from solid

models. This text

takes a hands-on

exercise intensive

approach to all the

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Creo Simulate 2
important Finite
Element Analysis
techniques and
concepts. This
textbook contains a
series of twelve
tutorial style
lessons designed to
introduce
beginning FEA
users to Creo
Simulate. The basic
premise of this

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book is the more designs you create using Creo Simulate, the better you learn the software. With this in mind, each lesson introduces a new set of commands and concepts, building on previous lessons. The eleven lessons

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in this tutorial
introduce you to
the design
capabilities of Creo
Parametric 5.0. The
tutorial covers the
major concepts and
frequently used
commands
required to advance
from a novice to an
intermediate user
level. Major topics

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include part and
assembly creation,

and creation of

engineering

drawings. Also

illustrated are the

major functions

that make Creo

Parametric a

parametric solid

modeler. Although

the commands are

presented in a click-

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by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the

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overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning

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and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the “ debugging ”

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phase of model
creation. At the end
of each lesson is a
short quiz
reviewing the new
topics covered in
that chapter.

Following the quiz
are several simple
"exercise" parts that
can be created
using new
commands taught

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in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and finally assembled at the end.

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- Written for first time FEA and Creo Simulate users •
- Uses simple examples with step-by-step tutorials •
- Explains the relation of commands to the overall FEA philosophy •
- Both 2D and 3D problems are

Get Free Introduction To Creo Simulate 2 covered Creo Simulate 8.0

Tutorial introduces new users to finite element analysis using Creo Simulate and how it can be used to analyze a variety of problems. The tutorial lessons cover the major concepts and frequently used

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commands
required to
progress from a
novice to an
intermediate user
level. The
commands are
presented in a click-
by-click manner
using simple
examples and
exercises that
illustrate a broad

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range of the
analysis types that
can be performed.
In addition to
showing the
command usage,
the text will explain
why certain
commands are
being used and,
where appropriate,
the relation of
commands to the

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overall Finite
Element Analysis
(FEA) philosophy
are explained.

Moreover, since
error analysis is an
important skill,
considerable time is
spent exploring the
created models so
that users will
become
comfortable with

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Inneo

the “ debugging ”
phase of modeling.
This textbook is
written for first-
time FEA users in
general and Creo
Simulate users in
particular. After a
brief introduction
to finite element
modeling, the
tutorial introduces
the major concepts

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behind the use of
Creo Simulate to
perform Finite
Element Analysis of
parts. These include
modes of
operation, element
types, design
studies (analysis,
sensitivity studies,
organization), and
the major steps for
setting up a model

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(materials, loads, constraints, analysis type), studying convergence of the solution, and viewing the results.

Both 2D and 3D

problems are

covered. This

tutorial deals

exclusively with

operation in

integrated mode

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Parametric. It is suitable for use with both Releases 8.0 of Creo Simulate. The tutorials consist of the following:

- 2 lessons on general introductory material
- 2 lessons introducing the basic

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Creo Simulate 2

operations in Creo

Simulate using solid

models • 4 lessons

on model

idealizations (shells,

beams and frames,

plane stress, etc) •

1 lesson on

miscellaneous

topics • 1 lesson

on steady and

transient thermal

analysis Table of

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Introduction To
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Contents 1.

Introduction to FEA

2. Finite Element
Analysis with Creo
Simulate 3. Solid

Models Part 1:

Standard Static

Analysis 4. Solid

Models Part 2:

Design Studies,

Optimization,

AutoGEM Controls,

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concepts, building
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features,
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and approach of

the previous

tutorial have been

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and the text picks

up right where the

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The material

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what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps, round sets, draft

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Advanced Tutorial
consists of eight
lessons. A

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continuing theme throughout the lessons is the creation of parts for a medium-sized modeling project. The project consists of a small three-wheeled utility cart. Project parts are given at the end of each

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lesson that utilize
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functions
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in that lesson.
Final assembly is
performed in the
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designers.

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this book describe the design process. The meat of this text, learning the basic Creo Parametric software, is found in Chapters 3 through 6.

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commands are
being used
and, where
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the relation
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Finite Element

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Analysis (FEA) philosophy are explained.

Moreover, since error analysis is an important skill, considerable time is spent exploring the created models

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written for
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users in
general and
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users in
particular.
After a brief
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modeling, the
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introduces the
major concepts
behind the use
of Creo
Simulate to
perform Finite
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Analysis of
parts. These
include modes

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of operation,
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element types,
design studies
(analysis,
sensitivity
studies,
organization),
and the major
steps for
setting up a
model
(materials,

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loads,
constraints,
analysis
type),
studying
convergence of
the solution,
and viewing
the results.
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tutorial deals
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exclusively
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mode with Creo
Parametric. It
is suitable
for use with
both Releases
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Simulate. The
tutorials

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Mechanism, a
module of the
PTC Creo
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software
family, which
supports
modeling and
analysis (or
simulation) of
mechanisms in
a virtual

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(computer)
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Capabilities
in Mechanism
allow users to
simulate and
visualize
mechanism
performance.

Capabilities
in Mechanism
allow users to

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performance.
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Mechanism
early in the
product
development
stage could
prevent costly
redesign due

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to design
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defects found
in the
physical
testing phase;
therefore,
contributing
to a more cost
effective,
reliable, and
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development
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process. The
book is
written
following a
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learning
approach and
covers the
major concepts
and frequently
used commands

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required to
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advance

readers from a
novice to an
intermediate
level. Basic
concepts
discussed
include: model
creation, such
as body and
joint

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definitions;
analysis type
selection,
such as static
(assembly)
analysis,
kinematics and
dynamics; and
results
visualization.
The concepts
are introduced

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using simple,
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yet realistic,
examples.

Verifying the
results
obtained from
computer
simulation is
extremely
important. One
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features of

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this textbook
is the
incorporation
of theoretical
discussions
for kinematic
and dynamic
analyses in
conjunction
with
simulation
results

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obtained using
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Mechanism. The
theoretical
discussions
simply support
the
verification
of simulation
results rather
than providing
an in-depth
discussion on

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the subjects
of kinematics
and dynamics.
Designing with
Creo
Parametric 7.0
provides the
high school
student,
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engineer with
a basic
introduction
to engineering
design while
learning the
3D modeling
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from PTC. The
topics are
presented in
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format with
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the end of
each chapter
to reinforce
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richly

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discusses
design intent
while you are
learning Creo
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the same time,
it shows how
knowledge
covered in
basic
engineering
courses such

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as statics,
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dynamics,
strength of
materials, and
design of
mechanical
components can
be applied to
design. You do
not need an
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degree nor be

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working toward
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a degree in
engineering to
use this
textbook.

Although FEA
(Finite
Element
Analysis) is
used in this
textbook, its
theory is not

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covered. The
Inneo
first two
chapters of
this book
describe the
design
process. The
meat of this
text, learning
the basic Creo
Parametric
software, is

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found in
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Chapters three
through six.
Chapters
seven, eight,
and 12 deal
with
dimensioning
and
tolerancing an
engineering
part. Chapters

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nine and ten
deal with
assemblies and
assembly
drawings.
Chapter 11
deals with
family tables
used when
similar parts
are to be
designed or

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used. Chapter

13 is an
introduction
to Creo
Simulate and
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**Creo
Inneo
Parametric 4.0**

The purpose of this book is to introduce the reader to 3D CAD/CAM modelling using Creo™ Parametric (Creo) software. This concise textbook consists of ten lessons covering the basics in Part and Assembly Modelling,

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Mould Design, NC
Simulation, and
Engineering
Drawings. Each
lesson provides
essential knowledge
and guides the user
through the process
of performing a
practical exercise or
task. The modelling
philosophy,
implementation of
corresponding

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features, and commands behind each exercise are explained and presented in a step-by-step manner. The material is richly illustrated with screenshots and icons from the software interface to facilitate the learning process. Suitable for beginners and

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Intermediate users,
CAD/CAM with Creo

Parametric enables

the reader to make a
quick start in learning

how to use complex
3D CAD/CAM

software such as

Creo in engineering
design and

manufacturing. The

aim is to develop an
understanding of the

main modelling

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principles and software tools as a basis for independent learning and solving more complex engineering problems. Designing with Creo Parametric 9.0 provides the high school student, college student, or practicing engineer with a basic introduction to

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engineering design
while learning the 3D
modeling Computer-
Aided Design

software called Creo
Parametric from PTC.

The topics are
presented in tutorial
format with exercises
at the end of each
chapter to reinforce
the concepts covered.
It is richly illustrated
with computer screen

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shots throughout.

Above all, this text is designed to help you expand your creative talents and

communicate your ideas through the graphics language.

Because it is easier to learn new information if you have a reason for learning it, this textbook discusses design intent while

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you are learning Creo Parametric. At the same time, it shows how knowledge covered in basic engineering courses such as statics, dynamics, strength of materials, and design of mechanical components can be applied to design. You do not need an engineering degree

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nor be working toward a degree in engineering to use this textbook.

Although FEA (Finite Element Analysis) is used in this textbook, its theory is not covered. The first two chapters of this book describe the design process. The meat of this text, learning the basic Creo Parametric

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software, is found in Chapters three through six. Chapters seven, eight, and 12 deal with dimensioning and tolerancing an engineering part. Chapters nine and ten deal with assemblies and assembly drawings. Chapter 11 deals with family tables used when

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similar parts are to be designed or used.

Chapter 13 is an introduction to Creo Simulate and FEA.

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 4.0. The tutorial covers the major concepts and frequently used commands required

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to advance from a
novice to an

intermediate user

level. Major topics

include part and

assembly creation,

and creation of

engineering drawings.

Also illustrated are the

major functions that

make Creo

Parametric a

parametric solid

modeler. Although the

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commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply

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knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought.

Moreover, since error recovery is an important skill,

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considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the “debugging” phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter.

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Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are introduced with the early lessons and

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finally assembled at
the end.

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 1.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user

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level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the major functions that make Creo Parametric a parametric solid modeler. These topics are further demonstrated in the video files that come

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Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage, to explain why certain commands are being used and the relation of feature selection and construction to

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the overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought.

Moreover, since error

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new topics covered in that chapter.

Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of several parts that are

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introduced with the
early lessons and
finally assembled at
the end.

The purpose of Creo
Parametric 7.0
Advanced Tutorial is
to introduce you to
some of the more
advanced features,
commands, and
functions in Creo
Parametric. Each
lesson concentrates

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on a few of the major topics and the text attempts to explain the "why's" of the commands in addition to a concise step-by-step description of new command sequences. This book is suitable for a second course in Creo Parametric and for users who understand the

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features already covered in Roger Toogood's Creo Parametric Tutorial.

The style and approach of the previous tutorial have been maintained from the previous book and the text picks up right where the last tutorial left off. The material covered in this tutorial represents an

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overview of what is felt to be the most commonly used and important functions. These include customization of the working environment, advanced feature creation (sweeps, round sets, draft and tweaks, UDFs, patterns and family tables), layers, Pro/PROGRAM, and

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advanced drawing
and assembly

functions. Creo
Parametric 7.0

Advanced Tutorial

consists of eight
lessons. A continuing
theme throughout the
lessons is the creation
of parts for a medium-
sized modeling
project. The project
consists of a small
three-wheeled utility

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cart. Project parts are given at the end of each lesson that utilize functions presented earlier in that lesson. Final assembly is performed in the last lesson.

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Mechanism 3.0 is
designed to help you
become familiar with
Mechanism, a module

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of the PTC Creo
Parametric software
family, which
supports modeling
and analysis (or
simulation) of
mechanisms in a
virtual (computer)
environment.

Capabilities in
Mechanism allow
users to simulate and
visualize mechanism

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Capabilities in Mechanism allow users to simulate and visualize mechanism performance. Using Mechanism early in the product development stage could prevent costly redesign due to design defects found in the physical testing

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phase; therefore,
contributing to a
more cost effective,
reliable, and efficient
product development
process. The book is
written following a
project-based
learning approach
and covers the major
concepts and
frequently used
commands required

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to advance readers
from a novice to an
intermediate level.

Basic concepts
discussed include:
model creation, such
as body and joint
definitions; analysis
type selection, such as
static (assembly)
analysis, kinematics
and dynamics; and
results visualization.

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The concepts are introduced using simple, yet realistic, examples. Verifying the results obtained from computer simulation is extremely important. One of the unique features of this textbook is the incorporation of theoretical

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discussions for
kinematic and
dynamic analyses in
conjunction with
simulation results
obtained using
Mechanism. The
theoretical
discussions simply
support the
verification of
simulation results
rather than providing

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discussion on the

subjects of kinematics

and dynamics.

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designers. Theoretical

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also introduced as
they are needed to

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operations. The

primary emphasis of

the text is placed on

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and procedures of

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in performing Linear
Statics Stress
Analysis; but the
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procedure is covered.
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to guide you from

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takes a hands-on

exercise intensive

approach to all the

important Finite

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techniques and

concepts. This

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series of twelve

tutorial style lessons

designed to introduce

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to Creo Simulate. The

basic premise of this

book is the more

designs you create

using Creo Simulate,

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the better you learn
the software. With
this in mind, each
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and concepts,
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this tutorial introduce
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capabilities of Creo
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illustrated are the major functions that make Creo Parametric a parametric solid modeler. Although the commands are presented in a click-by-click manner, an effort has been made, in addition to showing/illustrating the command usage,

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to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy.

Simply knowing where commands can be found is only half the battle. As is pointed out numerous times in the text,

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creating useful and effective models of parts and assemblies requires advance planning and forethought.

Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In fact, some errors are

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intentionally induced so that users will become comfortable with the "debugging" phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple "exercise"

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parts that can be
created using new
commands taught in
that lesson. In

addition to these an
ongoing project
throughout the book
is also included. This
project consists of
several parts that are
introduced with the
early lessons and
finally assembled at

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the end. Who this book is for This book has been written specifically with students in mind. Typically, students enter their first CAD course with a broad range of abilities both in spatial visualization and computer skills. The approach taken here

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is meant to allow
accessibility to
persons of all levels.

These lessons,
therefore, were
written for new users
with no previous
experience with CAD,
although some
familiarity with
computers is
assumed.

The eleven lessons in

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this tutorial introduce you to the design capabilities of Creo Parametric 6.0. The tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and

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assembly creation,

and creation of

engineering

drawings. Also

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considerable time is spent exploring the created models. In fact, some errors are intentionally induced so that users will become comfortable with the “debugging” phase of model creation. At the end of each lesson is a short quiz reviewing the new topics

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covered in that chapter. Following the quiz are several simple "exercise" parts that can be created using new commands taught in that lesson. In addition to these an ongoing project throughout the book is also included. This project consists of

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in spatial visualization and computer skills. The approach taken here is meant to allow accessibility to persons of all levels. These lessons, therefore, were written for new users with no previous experience with CAD, although some

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familiarity with
computers is

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Creating assemblies
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(FEA) that are
important to
engineers and

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designers. Theoretical
aspects of finite

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Analysis; but the

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Inpedagogical fashion
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book is the more

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Tutorial introduces
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exercises that illustrate a broad range of the analysis types that can be performed. In addition to showing the command usage, the text will explain why certain commands are being used and, where appropriate, the relation of commands to the

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overall Finite
Element Analysis

*(FEA) philosophy
are explained.*

*Moreover, since
error analysis is an
important skill,
considerable time is
spent exploring the
created models so
that users will
become comfortable
with the
“debugging” phase*

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of modeling. This textbook is written for first-time FEA users in general and Creo Simulate users in particular. After a brief introduction to finite element modeling, the tutorial introduces the major concepts behind the use of Creo Simulate to perform Finite

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Element Analysis of parts. These include: modes of operation, element types, design studies (analysis, sensitivity studies, organization), and the major steps for setting up a model (materials, loads, constraints, analysis type), studying convergence of the

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solution, and
viewing the results.
Both 2D and 3D
problems are
treated. This tutorial
deals exclusively
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integrated mode
with Creo
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suitable for use with
both Releases 3.0 of
Creo Simulate.
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provides the high school student, college student, or practicing engineer with a basic introduction to engineering design while learning the 3D modeling Computer-Aided Design software called Creo Parametric from

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PTC. The topics are presented in tutorial format with exercises at the end of each chapter to reinforce the concepts covered. It is richly illustrated with computer screen shots throughout. Above all, this text is designed to help you expand your

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creative talents and
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Because it is easier
to learn new
information if you
have a reason for
learning it, this
textbook discusses
design intent while
you are learning
Creo Parametric. At
the same time, it

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***shows how
knowledge covered
in basic engineering
courses such as
statics, dynamics,
strength of
materials, and
design of
mechanical
components can be
applied to design.
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engineering degree
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*toward a degree in
engineering to use
this textbook.*

*Although FEA
(Finite Element
Analysis) is used in
this textbook, its
theory is not
covered. The first
two chapters of this
book describe the
design process. The
meat of this text,
learning the basic*

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Creo Parametric software, is found in Chapters 3 through 6. Chapters 7, 8, and 12 deal with dimensioning and tolerancing an engineering part. Chapters 9 and 10 deal with assemblies and assembly drawings. Chapter 11 deals with family tables

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used when similar parts are to be designed or used. Chapter 13 is an introduction to Creo Simulate and FEA. The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 3.0. The tutorial covers the major concepts and

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frequently used
commands required
to advance from a
novice to an
intermediate user
level. Major topics
include part and
assembly creation,
and creation of
engineering
drawings. Also
illustrated are the
major functions that
make Creo

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***Parametric a
parametric solid
modeler. These
topics are further
demonstrated in the
video files that come
with every book.
Although the
commands are
presented in a click-
by-click manner, an
effort has been
made, in addition to
showing/illustrating***

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the command usage, to explain why certain commands are being used and the relation of feature selection and construction to the overall part design philosophy. Simply knowing where commands can be found is only half the battle. As is

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pointed out numerous times in the text, creating useful and effective models of parts and assemblies requires advance planning and forethought. Moreover, since error recovery is an important skill, considerable time is spent exploring the created models. In

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fact, some errors are intentionally induced so that users will become comfortable with the “debugging” phase of model creation. At the end of each lesson is a short quiz reviewing the new topics covered in that chapter. Following the quiz are several simple

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***"exercise" parts that
can be created
using new
commands taught in
that lesson. In
addition to these an
ongoing project
throughout the book
is also included.
This project
consists of several
parts that are
introduced with the
early lessons and***

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*finally assembled at
the end. Who this*

book is for This

book has been

written specifically

with students in

mind. Typically,

students enter their

first CAD course

with a broad range

of abilities both in

spatial visualization

and computer skills.

The approach taken

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here is meant to allow accessibility to persons of all levels. These lessons, therefore, were written for new users with no previous experience with CAD, although some familiarity with computers is assumed. The tutorials in this textbook cover the

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Introduction to the

following topics:

*Introduction to the
program and its
operation
The
features used in part
creation
Modeling
utilities
Creating
engineering
drawings
Creating
assemblies and
assembly drawings*

- Uses step-by-step
tutorials designed
for novice users •*

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Explains not only how but also why commands are used

- Covers part and assembly creation, creating engineering drawings and parametric solid modeling***

The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 8.0. The

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tutorial covers the major concepts and frequently used commands required to advance from a novice to an intermediate user level. Major topics include part and assembly creation, and creation of engineering drawings. Also illustrated are the

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make Creo
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parametric solid
modeler. Although
the commands are
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by-click manner, an
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quiz reviewing the
new topics covered
in that chapter.
Following the quiz
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can be created
using new

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This project consists of several parts that are introduced with the early lessons and finally assembled at the end. Who this book is for This

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operation • The features used in part creation • Modeling utilities • Creating engineering drawings • Creating assemblies and assembly drawings
The eleven lessons in this tutorial introduce you to the design capabilities of Creo Parametric 2.0. The tutorial

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concepts and
frequently used
commands required
to advance from a
novice to an
intermediate user
level. Major topics
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and creation of
engineering
drawings. Also
illustrated are the

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showing/illustrating*

the command

usage, to explain

why certain

commands are

being used and the

relation of feature

selection and

construction to the

overall part design

philosophy. Simply

knowing where

commands can be

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early lessons and
finally assembled at
the end.*

*Designing with Creo
Parametric 7.0*

*Designing with Creo
Parametric 5.0*

*Designing with Creo
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*Tutorial and
Multimedia CD*

*Introduction to
Finite Element*

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***Analysis Using Creo
Simulate 4.0***

The primary
goal of
Introduction
to Finite
Element
Analysis Using
Creo Simulate
1.0 is to
introduce the
aspects of

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finite element
analysis (FEA)

that are

important to

the engineers

and designers.

Theoretical

aspects of

finite element

analysis are

also

introduced as

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they are
needed to help
better
understand the
operations.
The primary
emphasis of
the text is
placed on the
practical
concepts and
procedures of

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using Creo
Inneo
Simulate in
performing
Linear Statics
Stress
Analysis; but
the basic
modal analysis
procedure is
covered. This
text is
intended to be

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used as a
training guide
for both
students and
professionals.
This text
covers Creo
Simulate 1.0
and the
lessons
proceed in a
pedagogical

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fashion to
guide you from
constructing
basic truss
elements to
generating thr
ee-dimensional
solid elements
from solid
models. This
text takes a
hands-on

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exercise
intensive
approach to
all the
important
Finite Element
Analysis
techniques and
concepts. This
textbook
contains a
series of

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twelve
Inneo
tutorial style
lessons
designed to
introduce
beginning FEA
users to Creo
Simulate. The
basic premise
of this book
is the more
designs you

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create using
Inneo
Creo Simulate,
the Better you
learn the
software. With
this in mind,
each lesson
introduces a
new set of
commands and
concepts,
building on

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previous
lessons.
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Tutorial
Release 1.0 &
2.0
Cad/cam With
Creo
Parametric:
Step-by-step
Tutorial For
Versions 4.0,

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Parametric 6.0
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Analysis Using
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3.0
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to Finite

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Analysis Using
Creo Simulate
7.0