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The purpose of
Advanced Tutorial for
Creo Parametric is to
introduce you to some
of the more advanced
features, commands,
and functions in Creo
Parametric Releases
1.0 and 2.0. Each
lesson concentrates
on a few of the major
topics and the text
attempts to explain
the "why's" of the

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Commands in addition to a concise step-by-step description of new command sequences.

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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 lesson that utilize functions presented earlier in that lesson.

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3d Modeling Tutorial.

In this video, I have demonstrated a very simple 3d model. This is for those who know part modeling commands in Creo Parametric. I am not explaining "how extrude works" or

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Creo Parametric Help Center - PTC

In Creo Parametric,
the parametric part
modeling process
involves the following
steps: 1. Set up Units

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and Basic Datum

Geometry. 2.

Determine the type of
the base feature, the

first solid feature, of

the design. Note that

Extrude, Revolve, or

Sweep operations are

the most common

types of base

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In this video, you are going to create a 3d model which is a very good exercise for beginners as well as freshers in the field.

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In this video, I have
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simple 3d model. This
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part modeling
commands in Cre...

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