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Block This

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is a simple

block that

requires many of
the same

commands as the
Clamp. Using the
default datums
create the part.

Sketch the

protrusion on
datum FRONT.

Sketch the cut

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5.0 3d Modeling
Tutorial. In
this video, I
have*

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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
similar things.
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available as
version 7. Since

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was renamed Creo Parametric in 2011, PTC has worked massively on the overhaul of the user interface and usability. Creo is intuitive to use today without sacrificing functionality,

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robustness and
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have a modern
ribbon interface
at our disposal
that follows
very strongly
the philosophy
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and leverage
product designs
downstream
utilizing 2D
CAD, 3D CAD,
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The eleven
lessons in this

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tutorial
introduce you to
the design
capabilities of
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5.0. The
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the major
concepts and
frequently used
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required to
advance from a
novice to an

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Major topics
include part and
assembly
creation, and
creation of
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work by creating
a folder for
each project.
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start Creo
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should set the
working
directory to the
folder you plan
to work in. In
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instructed to
create a folder
and set that as
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extrude feature,
the direction of
the feature

determines if
the extrude will
add or remove
material from
the model.

Orienting
Components.

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Parametric's 3D
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component during
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