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of freedom consistent with a cylindrical joint between the block and the base and a revolute joint between the block and the conrod.

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Step 3: Part Design Module. Part design environment is used to create 3D models from the basic 2D sketches created in sketcher environment. Some of the commands in workbench explained below. PAD command. In most CAD software, the equivalent of this is called EXTRUDE, but in CATIA we call it PAD.

## **CATIA V5 Basic Tutorial | GrabCAD Tutorials**

Instructional Video: Intro to CATIA Assembly Mechanisms ETEC

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341 - Fall 2012 Created By: Daniel Burke and Yonatan Gebremedhin ... Universal joint - CATIA V5 Tutorial (part design, assembly ...

## **CATIA Assembly Mechanism Tutorial**

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