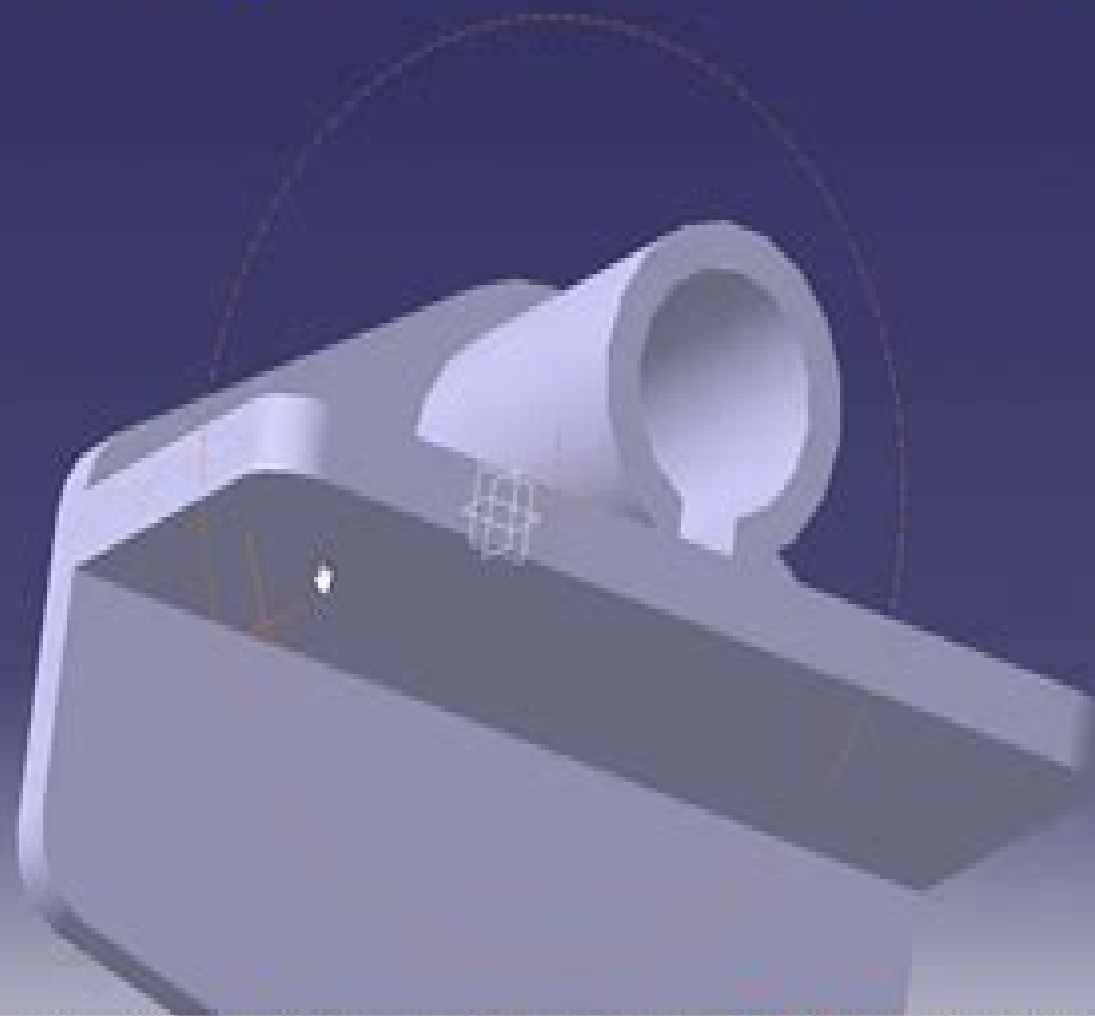




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Catia V5 Guide

Prof. Sham Tickoo



Catia V5 Guide:

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CATIA V5 Surface Design with Applications Jaecheol Koh, 2019-09-20 This textbook explains how to create models with freeform surfaces using CATIA V5. CATIA is a three dimensional CAD CAM CAE software developed by Dassault Syst ms France. This textbook is based on CATIA V5 6R2014. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. All files are in CATIA V5R20 so readers can open the files using later releases of CATIA V5. It is assumed that readers of this textbook are accustomed to the modeling tools and processes in how to construct solid models in CATIA V5. For basic modeling assembly and drafting techniques refer to the textbook written by the author. This textbook is suitable for anyone who are interested in learning how to create and use the freeform surface in constructing 3D models using CATIA V5.

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