

SHAM TICKOO CATIA DESIGNERS GUIDE

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Introduction

Starting a Drawing File

Drafting Workbench

Top View

Detail View

Isometric View

All Views

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Introduction

Customize toolbars

Create a base feature

Create a rib feature

Multisection solid feature

Offset from plane

Sketcher workbench

Multiset

Edges

Shell

Sketcher

Pocket Feature

Clear View

Pattern Pocket

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Is Catia Parametric?

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Start a New Part File

Sketcher Workbench

Snap to Point

Draw an Arc

Three-Point Arc Tool

Draw the Circle

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Multi Section Surface

Create the Multi-Section Surface

Trim Tool

Solidify the Model Using the Closed Surface Tool

Elbow Design I CATIA V5 Part Design I exercise - Elbow Design I CATIA V5 Part Design I exercise by

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Introduction to Catia

What is Catia

Catia GUI

2D sketching toolboxes

2D sketching

3D sketching toolboxes

3D sketching

Summary

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add the tangent relation between these lines

add the vertical relation

dimension it 35 millimeter

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