# TREINAMENTO DE CATIA V5 BÁSICO 1

20 horas

1 INTRODUÇÃO

2 SKETCH

3 DESIGN 3D

4 ASSEMBLY DESIGN

5 DRAFTING

LWT
SISTEMAS

# Introdução

01 Open a CATIA Session

02 Explore the CATIA User Interface

03 Design Intent



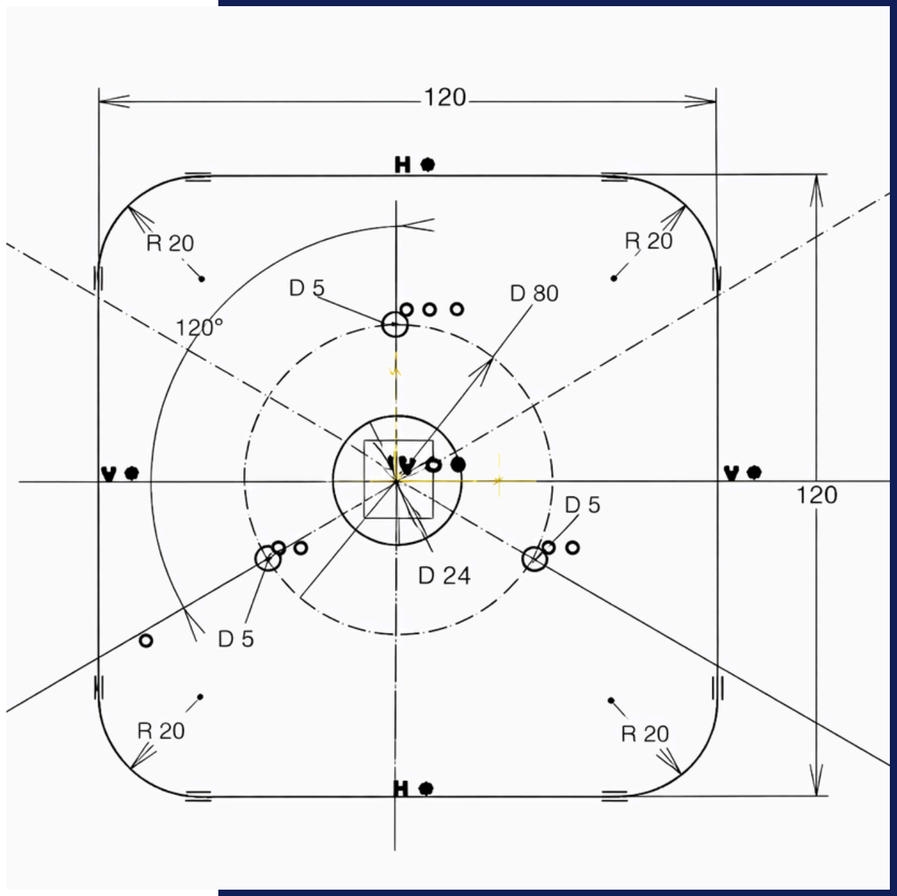

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# Sketch

01 Select an Appropriate Sketch Support

02 Create Sketched Geometry

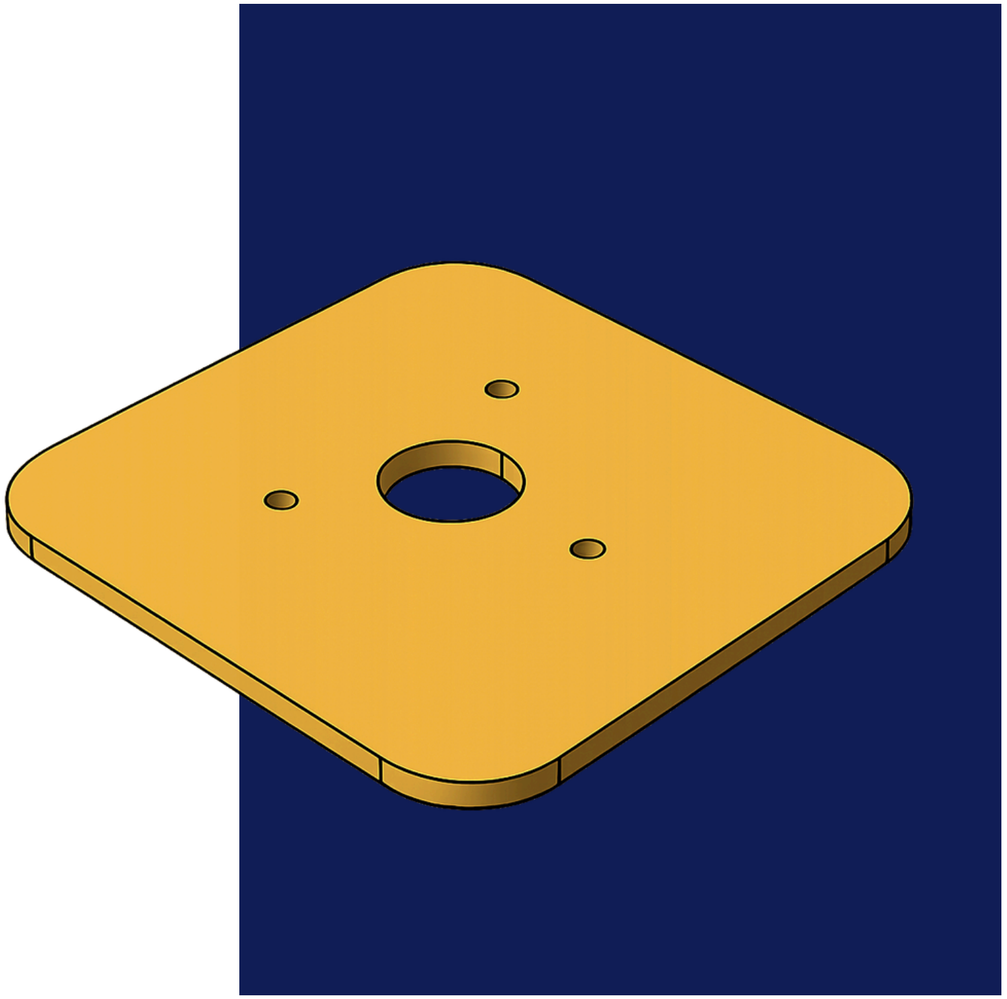
03 Constrain the sketch




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# Design 3D

01 Creating Pad an Pocket features
02 Creating Holes
03 Creating Fillets and Chanfers
04 Create Shaft and Groove features
05 Apply material
06 Measure the model



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# Assembly Design

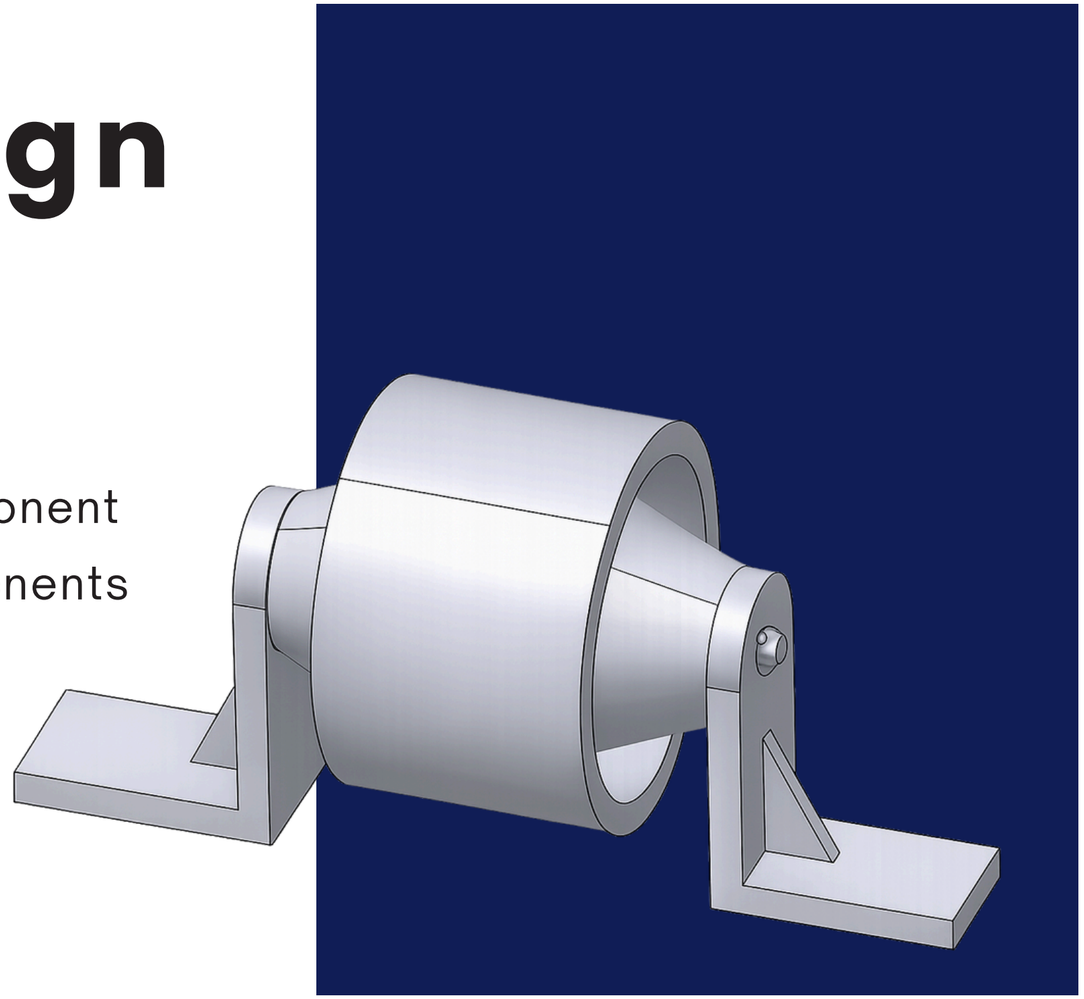
01 Create a new assembly

02 Adding Components

03 Manipulate the position of the component

04 Assembly and Fully constrain components

05 Save the Assembly



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# Drafting

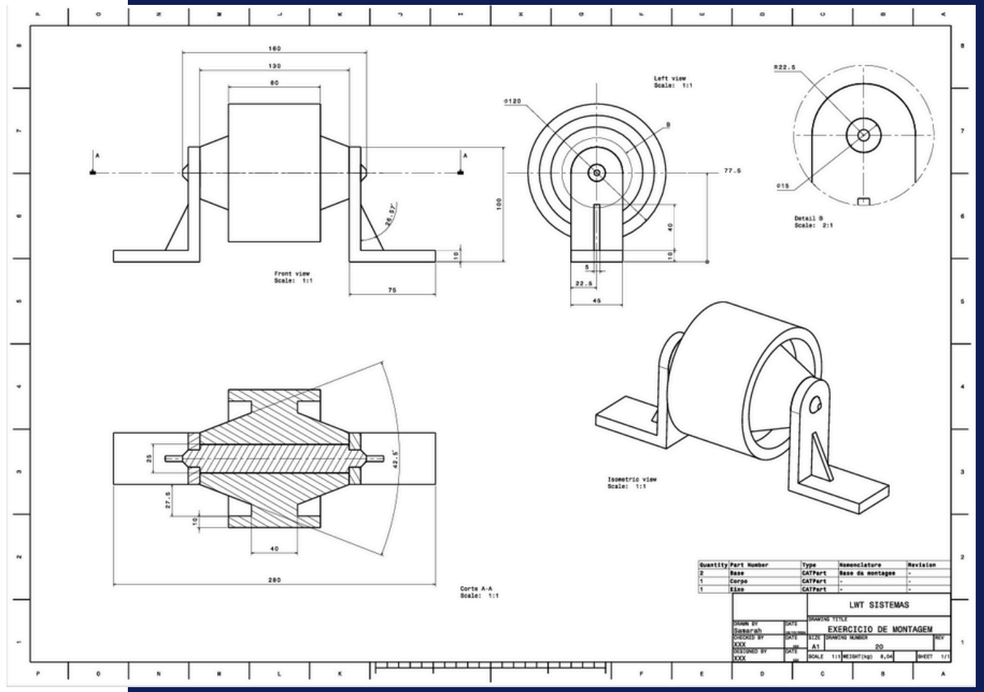
01 Create a new drawing

02 Create views

03 Create Dimensions an Annotations

04 Save the Drawing

05 Print the Drawing




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