# EE-100 Engineering Laboratory Module2: CAD

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Office Hours: Room 9-245A

Tuesday (1000-1100)

Thursday (1000-1100)

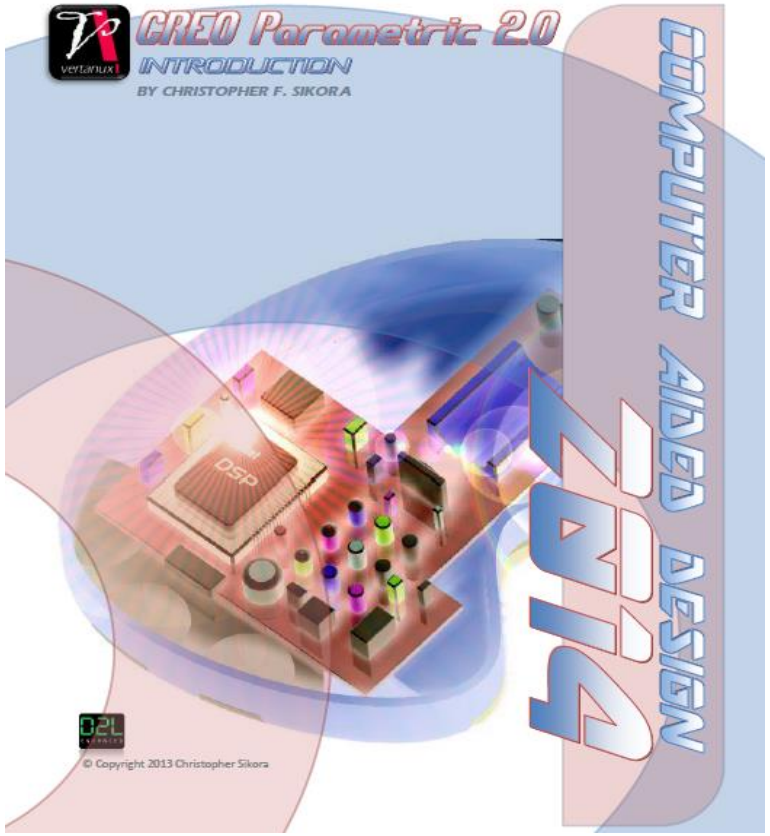
# CAD Module

**Learning Objective 1:** Create and interpret mechanical drawings
**Learning Objective 2:** Recall and demonstrate workshop/industrial safety practices.

- Week 1
  - Design methodology for scientists and engineers
  - Introduction to Engineering Drawing
  - Engineering Drawings
    - Standards, Types, Projections
  - **Lab Task 5:** Sketch orthographic projections of solid objects
- Week 2
  - Computer Aided Modeling
    - Intro to PTC Creo and its features
    - 2D sketching
    - Basics of 3D object modeling
  - **Lab Task 6**: 3D part modeling.
- Week 3
  - Advanced features of PTC Creo Parametric 2.0:
  - **Lab Task 7:** 3D part modeling
- Week 4
  - Assembly
  - **Lab Task 8:** Assembly task
- Week 5
  - **Lab Task 9:** Create parts and assembly drawings for a robotic hand (gripper)

# References

- Creo Parametric 2.0 Introduction
  - By Christopher F. Sikora



# CAD/CAM

- Computer-aided design (CAD) is the use of computer systems to assist in the design process for
  - Creation
  - Modification
  - Analysis
  - Optimization

# CAD/CAM (cont.)

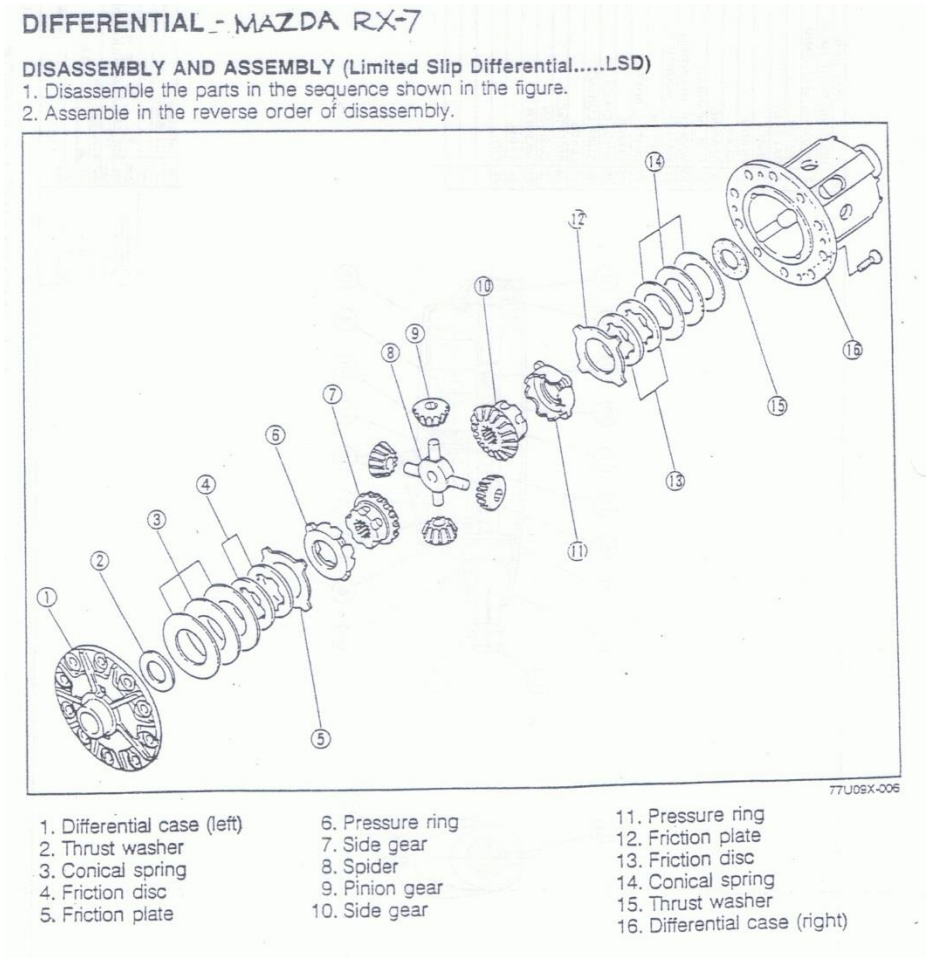
Computer-aided manufacturing (CAM) is the use of computer systems to *plan*, *manage*, and *control* the operations of a manufacturing plant through direct or indirect computer interface with plant's resources.

# Need for CAD/CAM

- Increases productivity of the designer
- Improves quality of the design
- Improves communications
- Creates a manufacturing database
- Create and test toolpaths and optimize them
- Helps in production scheduling and MRP models
- To have effective shop floor control

# Parts And Assembly

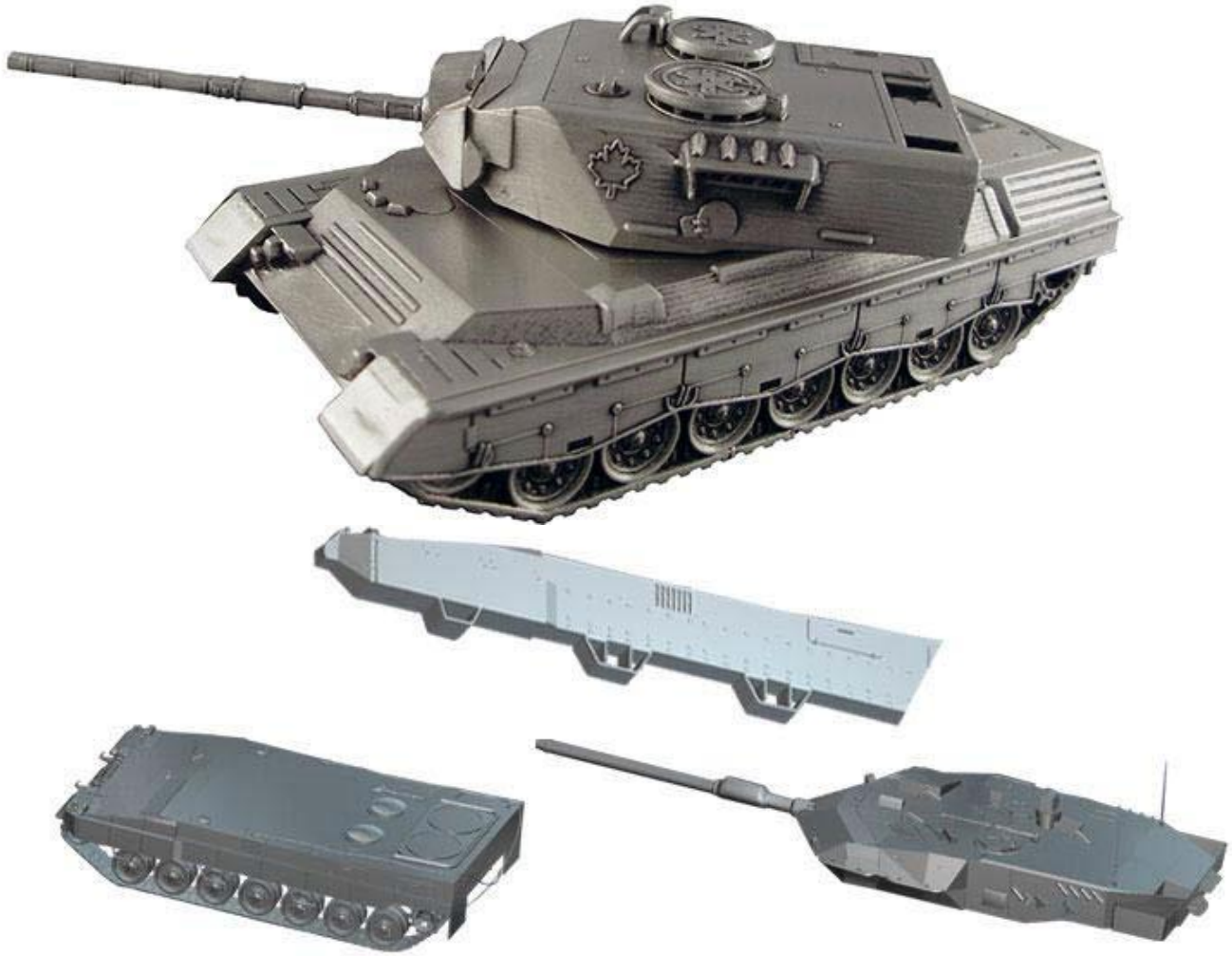
- **A part** is a model representing a single, continuous object e.g. screw, fan
- **An assembly** is a combination of multiple parts that form a larger system e.g. LED cooling system (shown below)




# Parts And Assembly (Cont.)

- **Example:**

Turret of the Canadian Leopard tank is a separate assembly so it can swivel around the base. The turret consists of eight individual parts.



# Features

- **Features are the basic building blocks we use to create an object.**
  - For example, a hole “knows” its shape and location and that fact that it has a negative volume.
- As you modify a feature, the entire object automatically updates after regeneration.

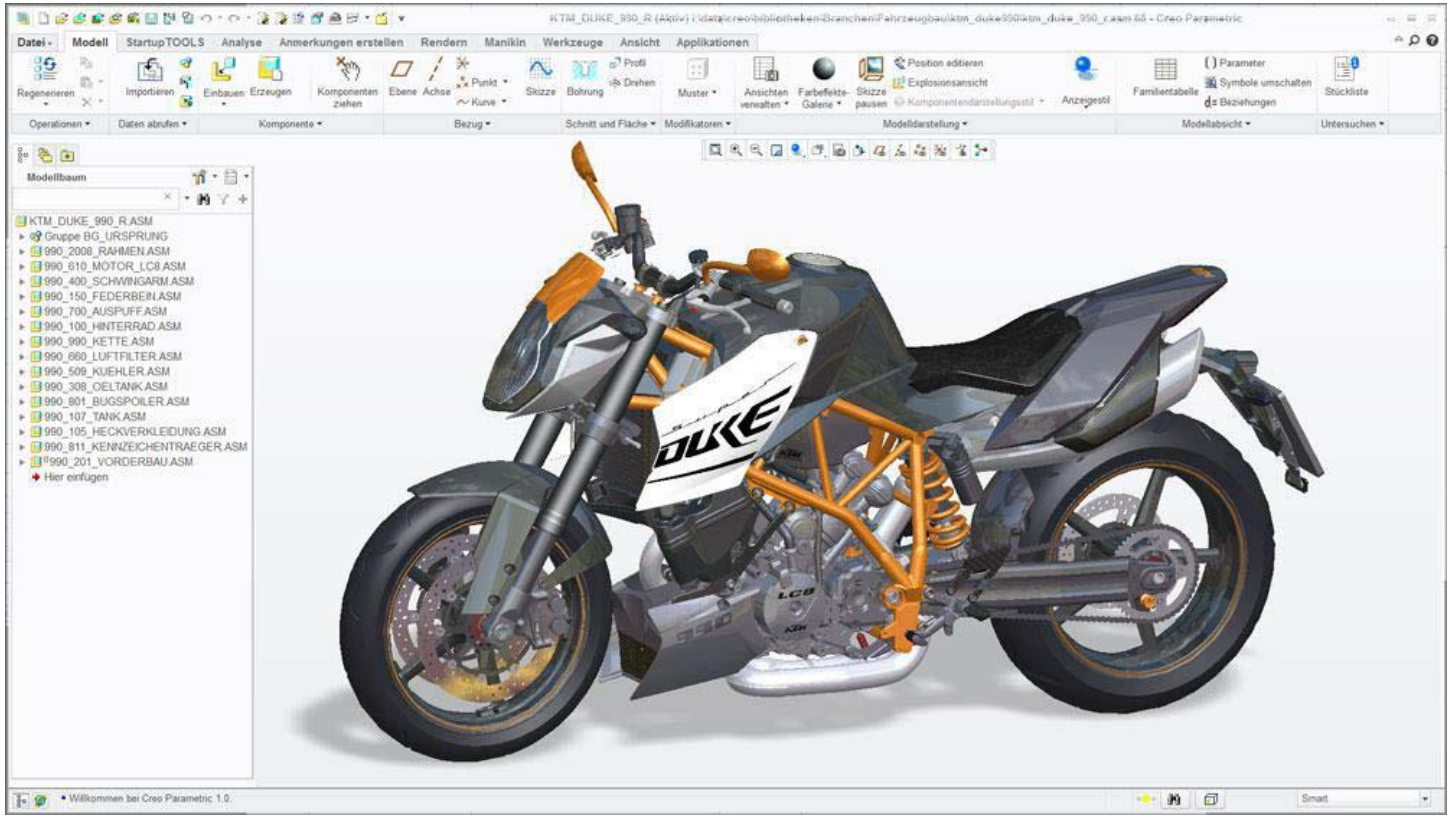
- The idea behind feature-based modeling is that the designer constructs an object so that it is composed of individual features that describe the way the geometry is supposed to behave if its dimensions change.

# Features (Cont.)

- Various types of features are used, such as base features, datum features, sketched features, and referenced features.
- The "chunks" of solid material from which we will construct our models are called **features**.

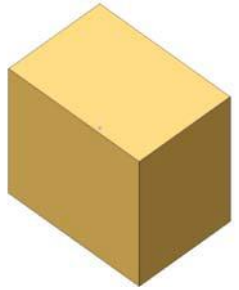
# PTC CREO PARAMETRIC 2.0

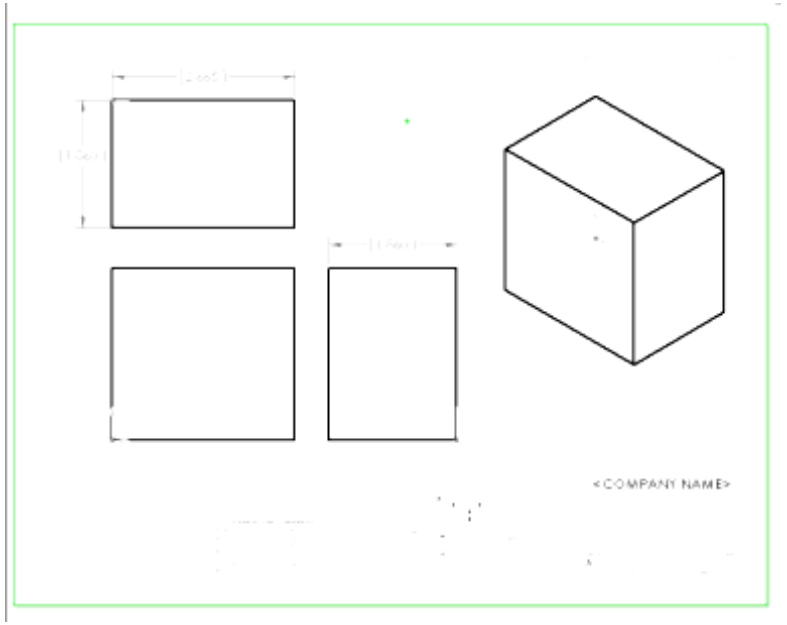
Suite of 2D and 3D product design software used to **create, analyze and view product designs**.

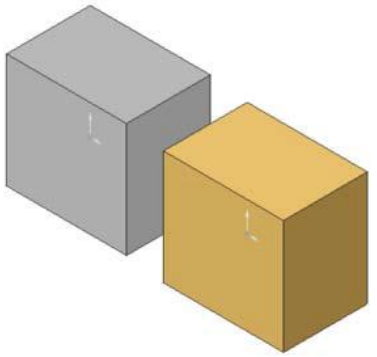


# Creo File Types

- ***Part*** (.prt)
  - Single part or volume
- ***Assembly*** (.asm)
  - Multiple parts in one file assembled
- ***Drawing*** (.drw)
  - The 2D layout containing views, dimensions, and annotations





# TWO APPROACHES TO DESIGN

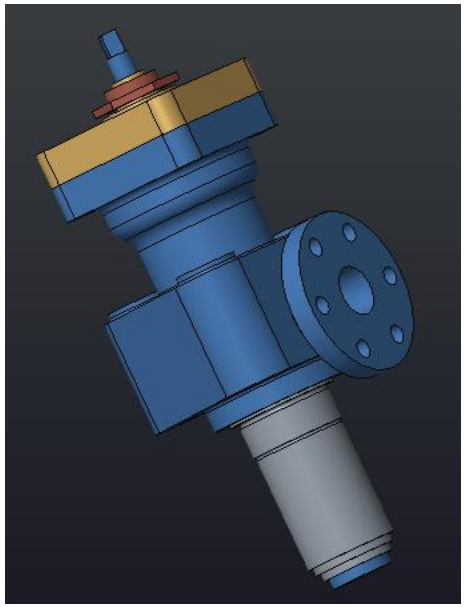
- BOTTOM UP DESIGN
- TOP DOWN DESIGN

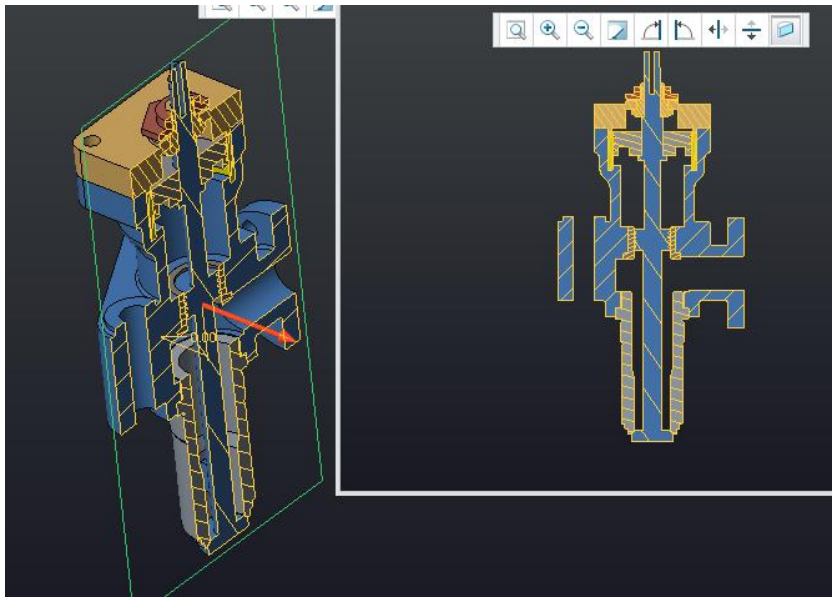
# BOTTOM-UP DESIGN

- Have the definition of parts in the system that we design
- Assemble every parts together to build the final assembly
- Create drawings for each components ,subassembly and final assembly

# PROCESS OF BOTTOM-UP DESIGN IN Creo

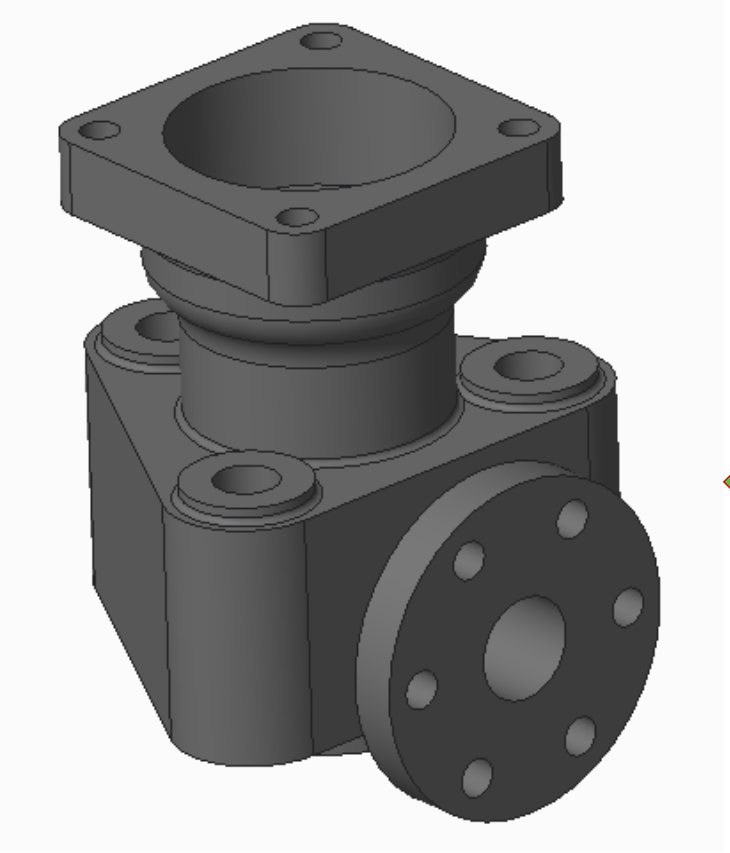
Individual components assembled together to create a final assembly





# PROCESS OF BOTTOM-UP DESIGN IN CREO

Drawings is referred to the  Component.
Hence a change in the Component reflects the drawing automatically



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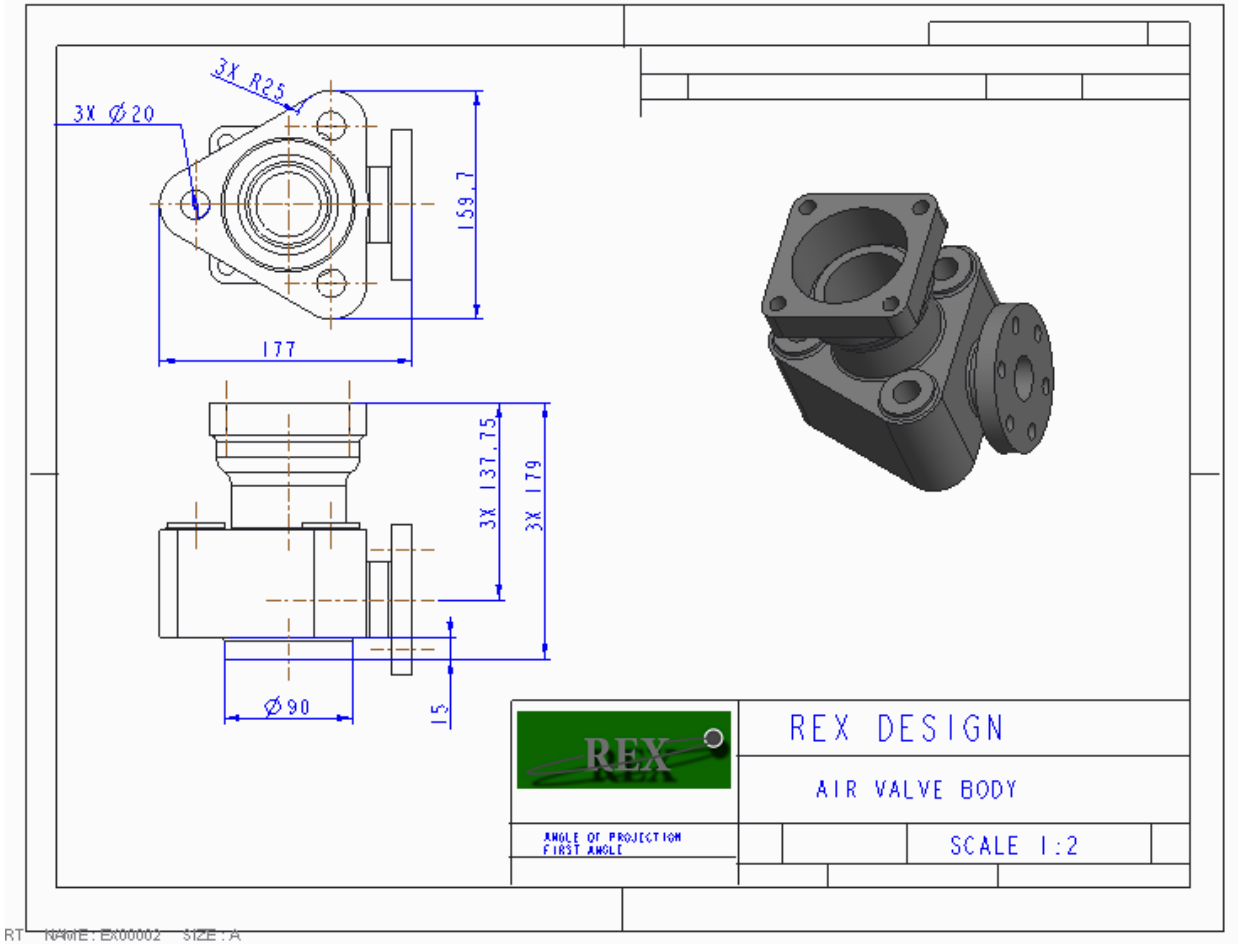
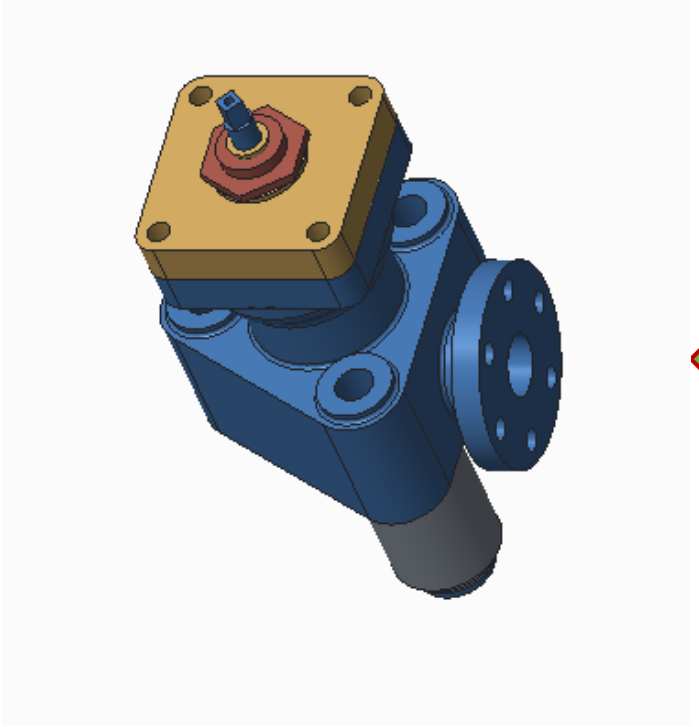



Also a component definition change in the drawing reflects the part too.

# PROCESS OF BOTTOM-UP DESIGN IN CREO

Drawings is referred to the  Assembly.
Hence a change in the Assembly reflects the drawing automatically



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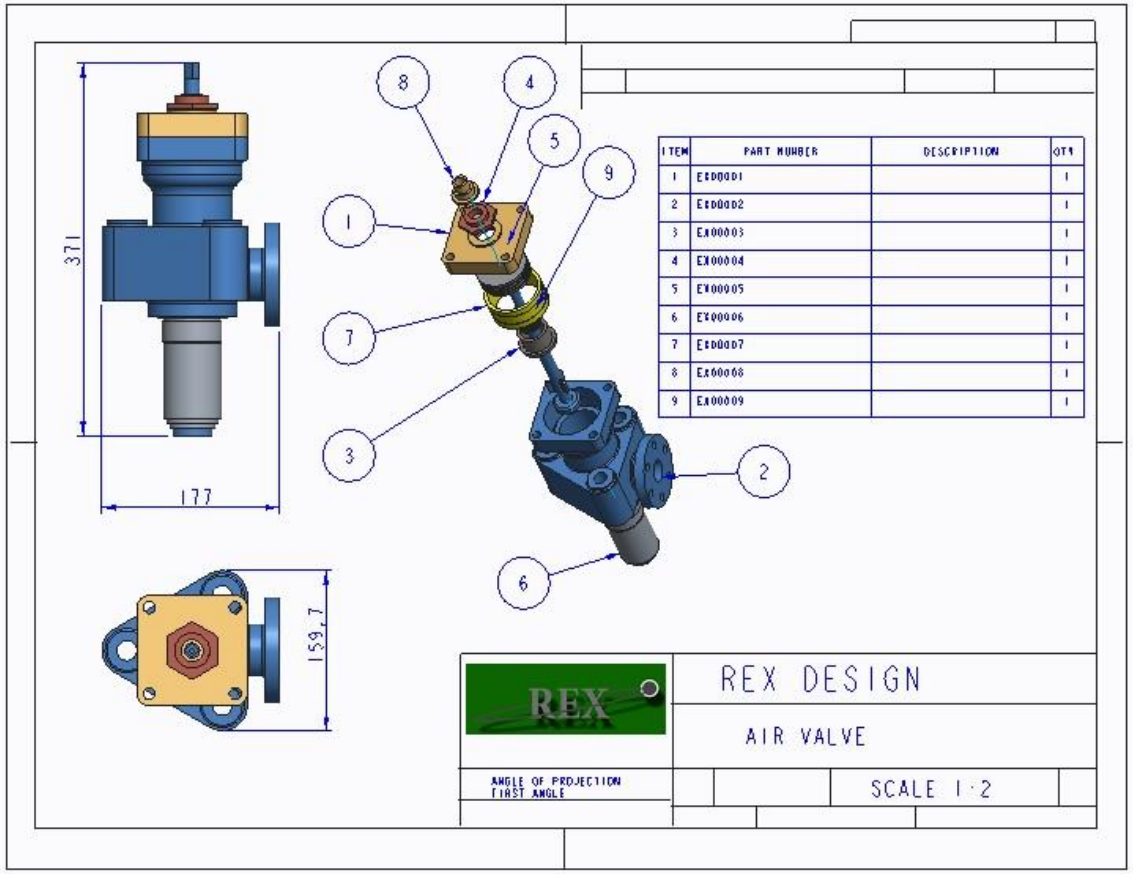
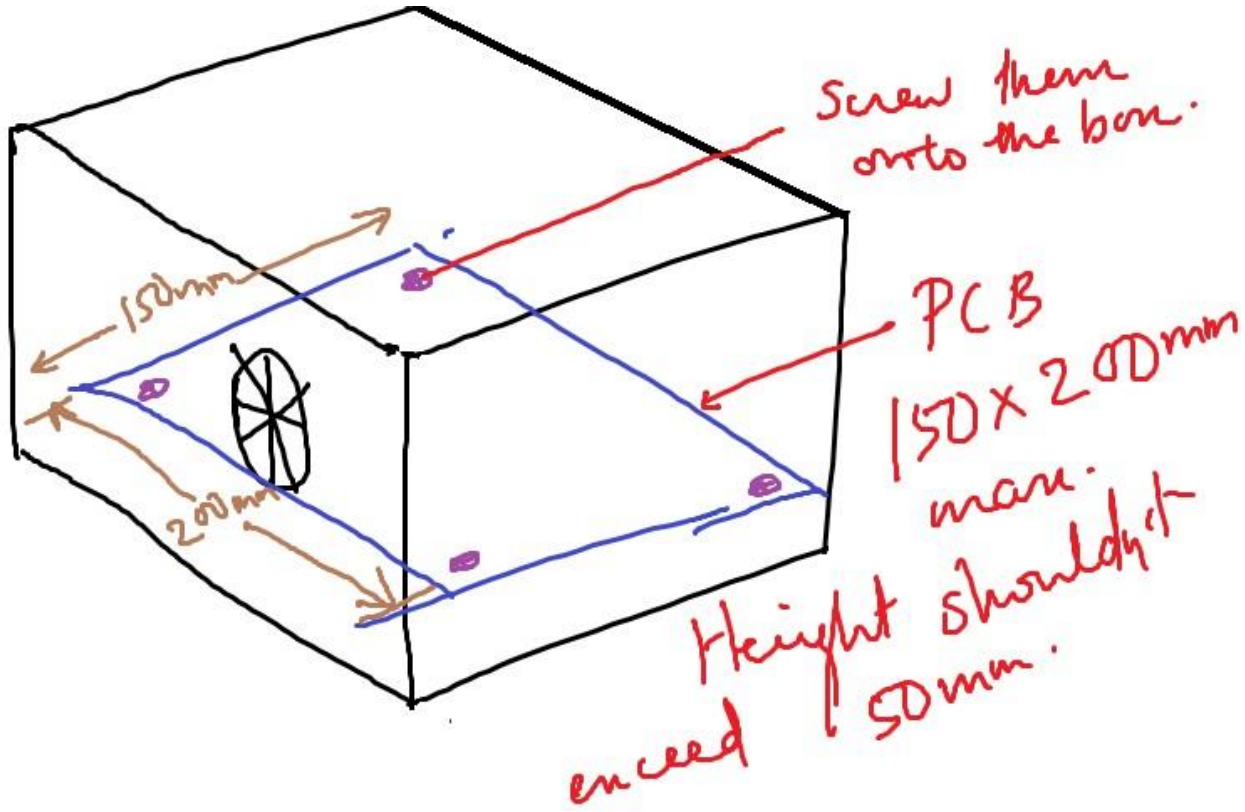



Also a component definition change in the drawing reflects the Assembly too.

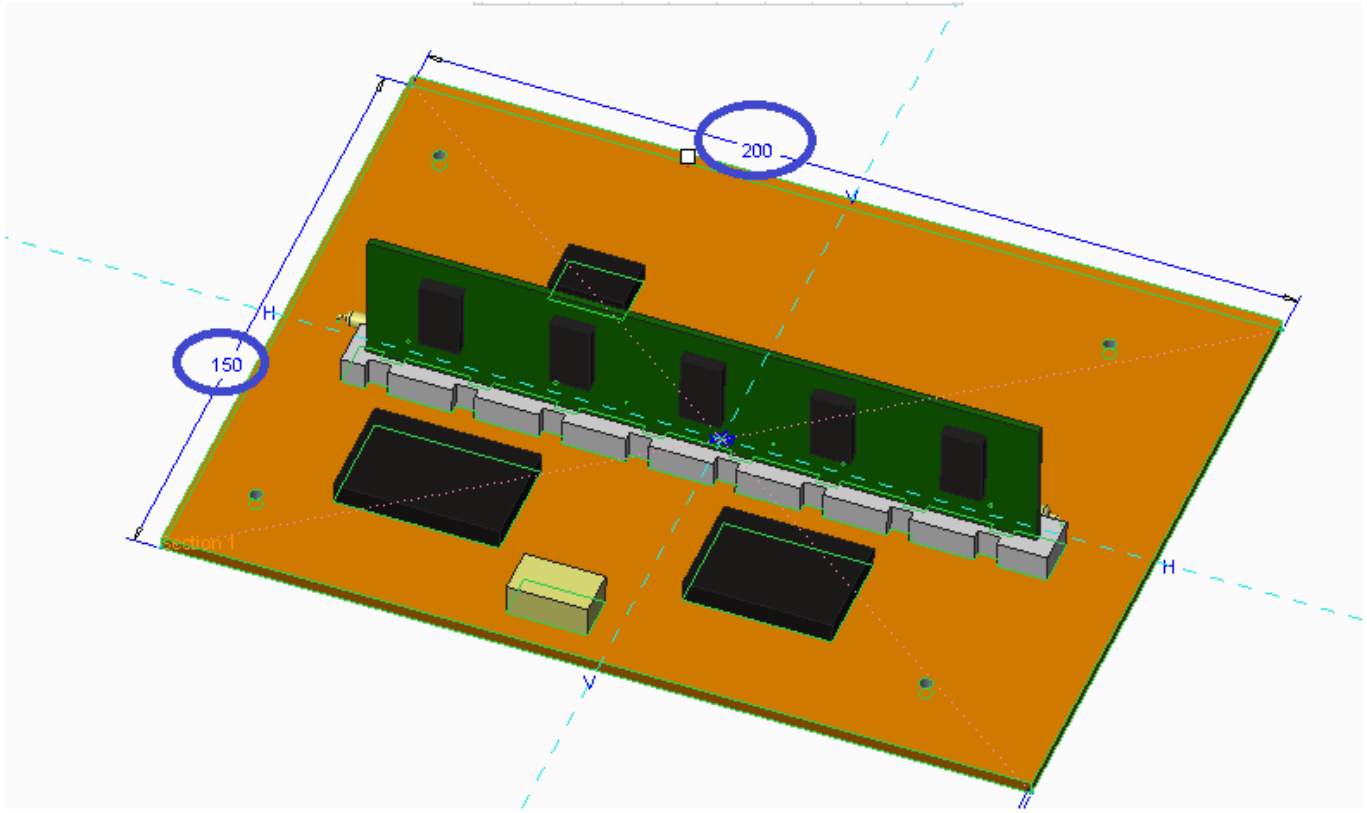
# PROCESS OF TOP-DOWN DESIGN IN CREO




Design objective here is to make a cabinet (enclosure) for the printed circuit board.

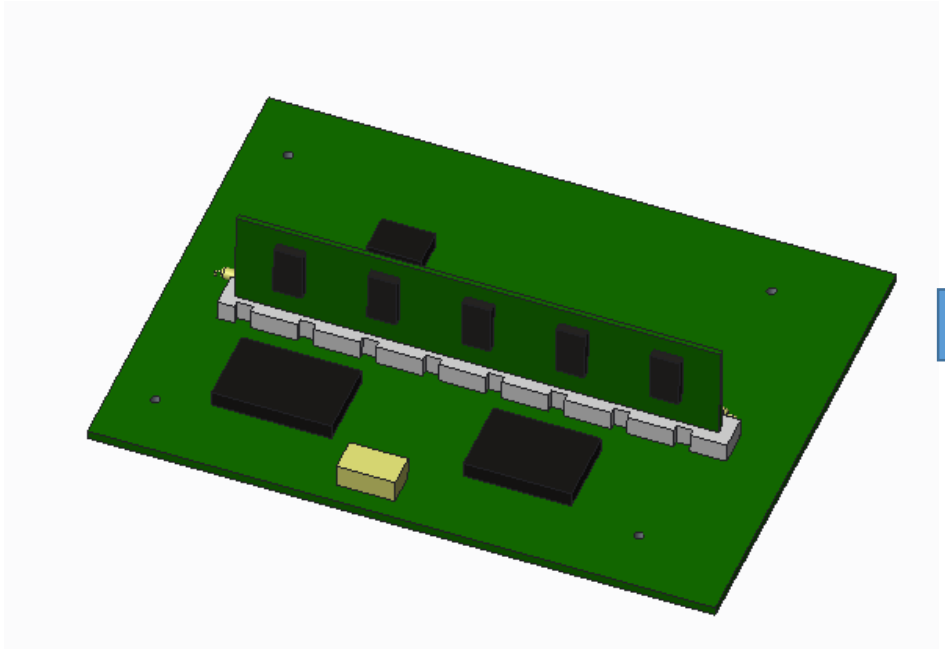
The inputs shall be a dummy PCB board design and other design parameters (For E.g.. the heat generated per hour etc.)

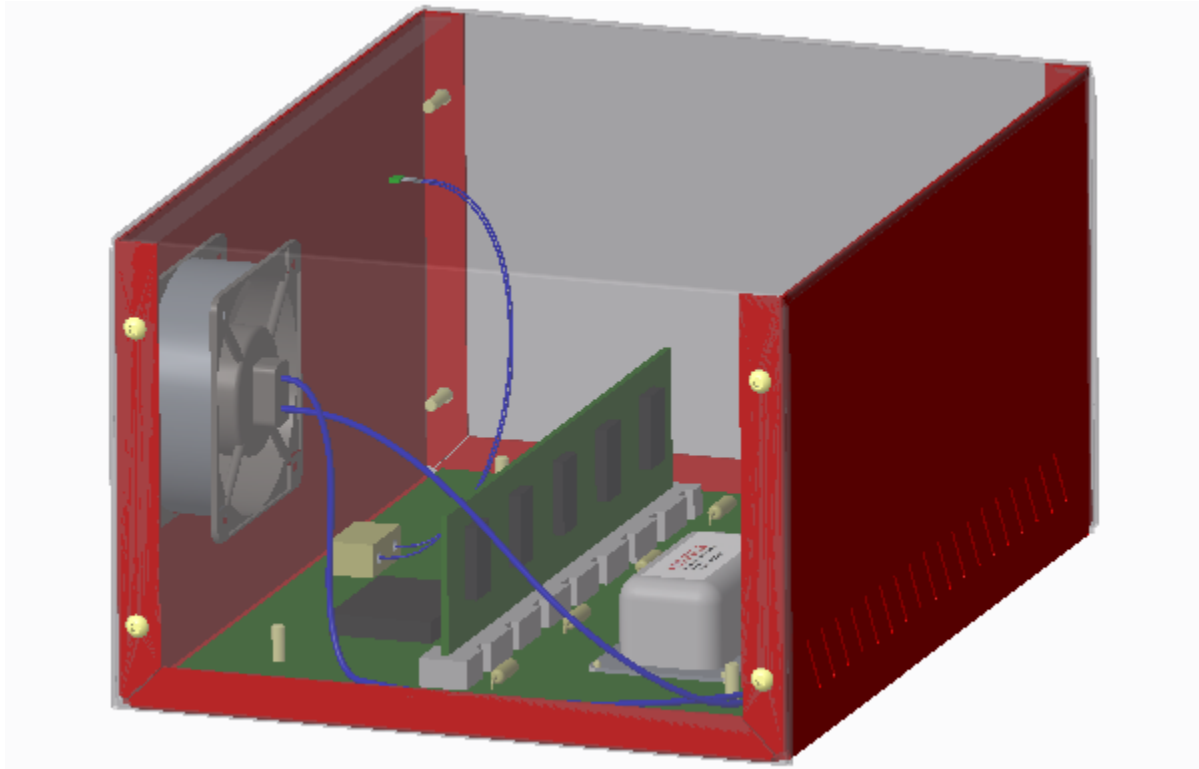
# PROCESS OF TOP-DOWN DESIGN IN CREO




Here we have the design input, the PCB
Shown the overall length X width. Height is not shown here which is important too.

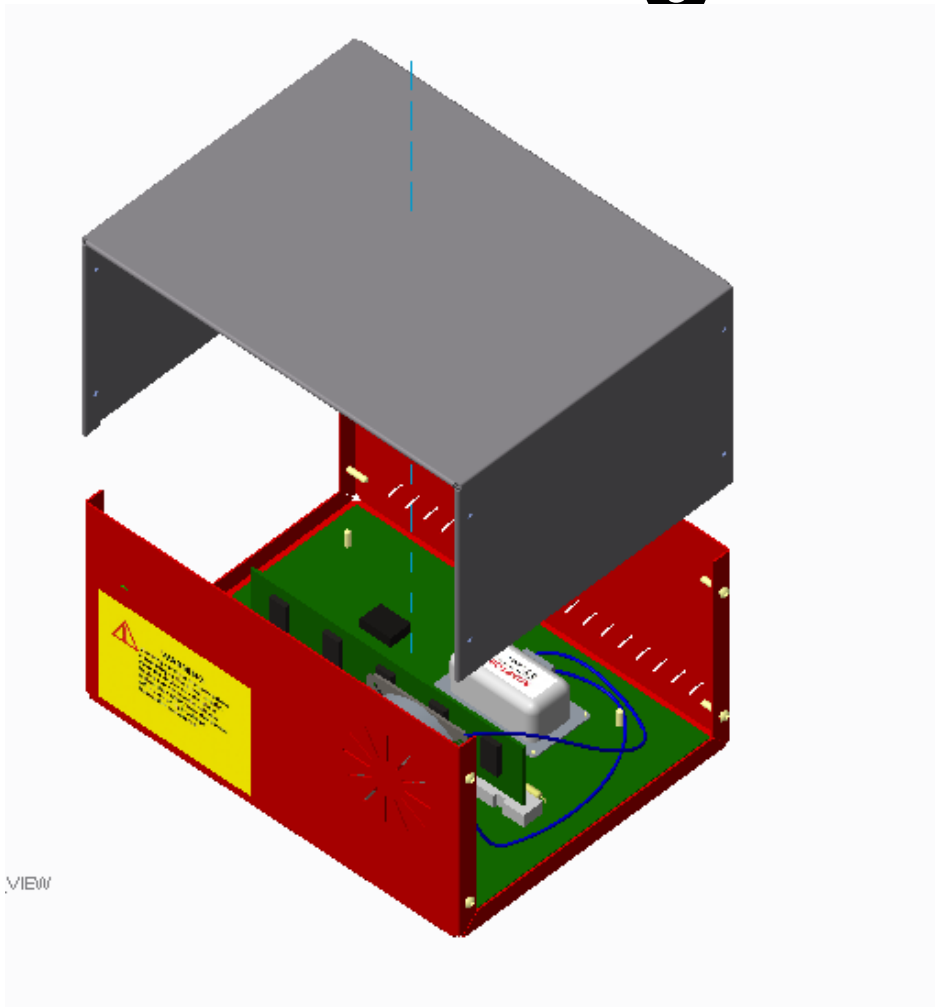
# PROCESS OF TOP-DOWN DESIGN IN CREO

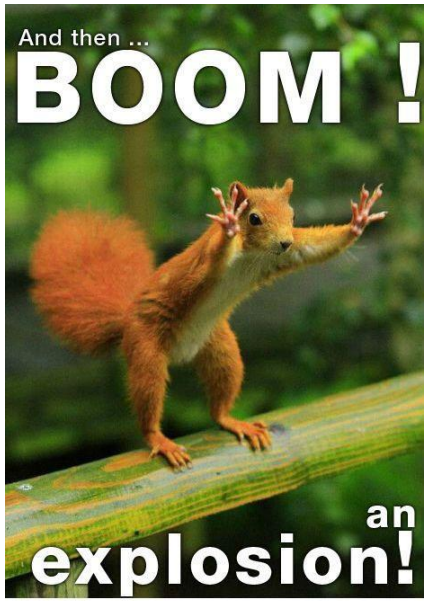




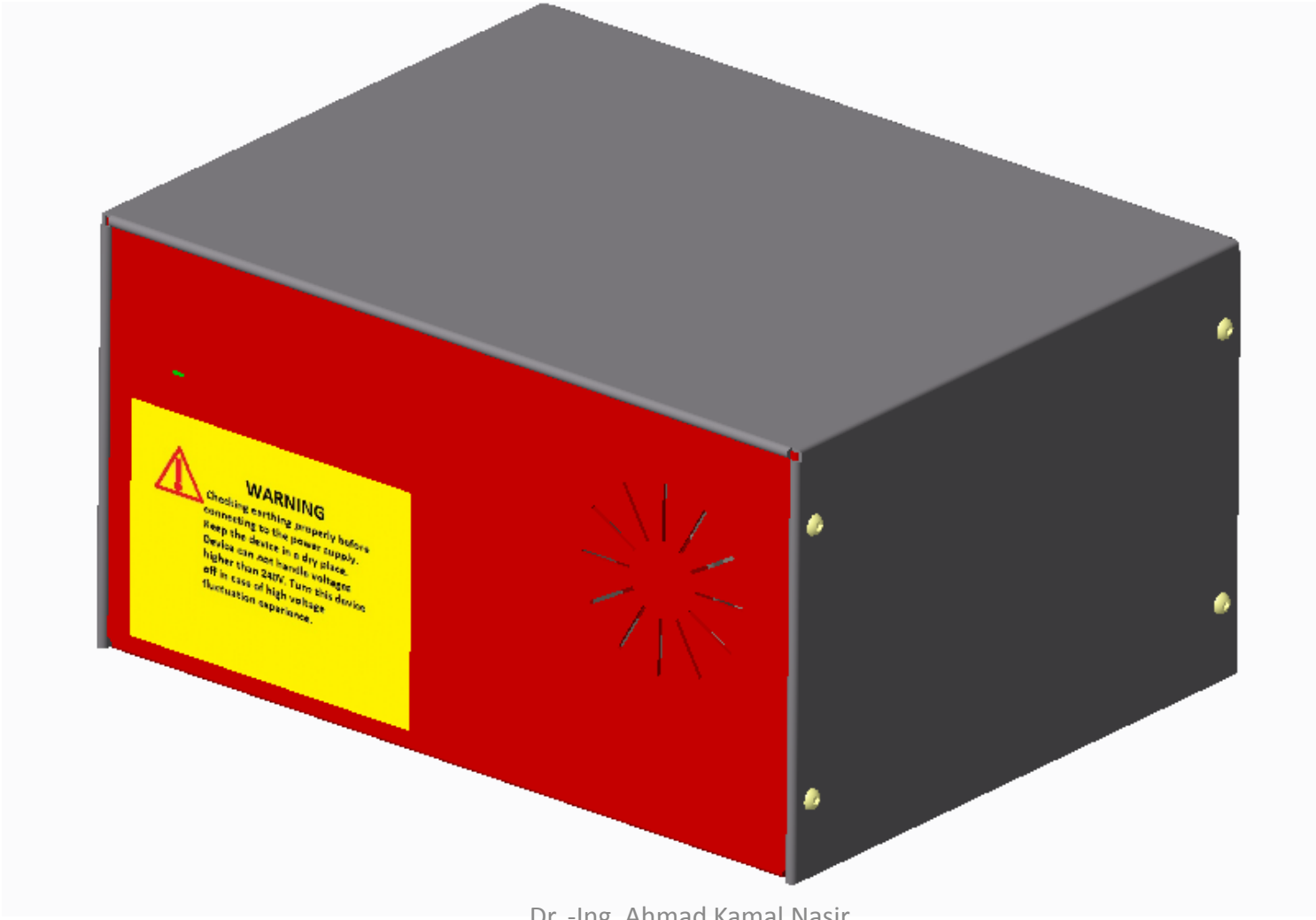
Shows the Top-Down approach in designing a cabinet from the inputs given.

# Exploded View showing the inside details

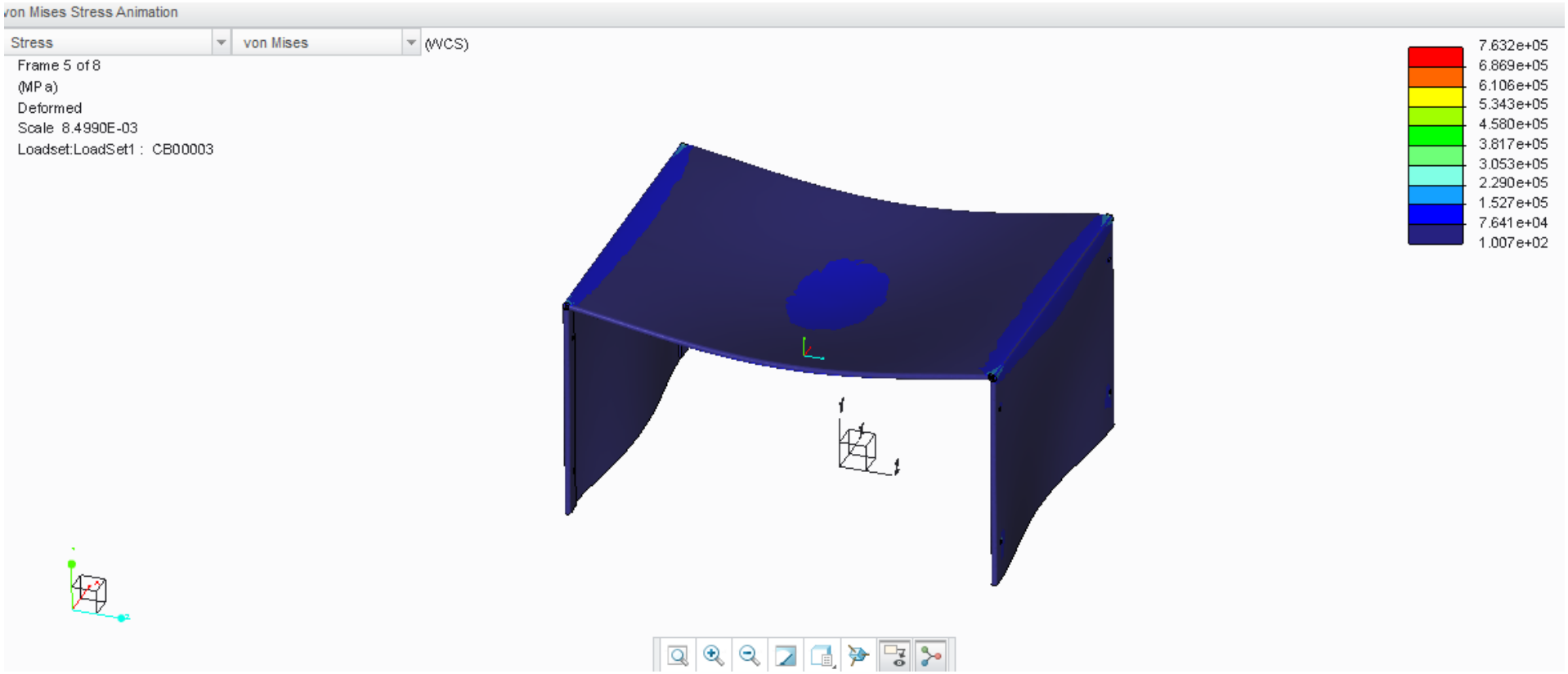




# Design-Iteration 1



# TESTING THE DESIGN

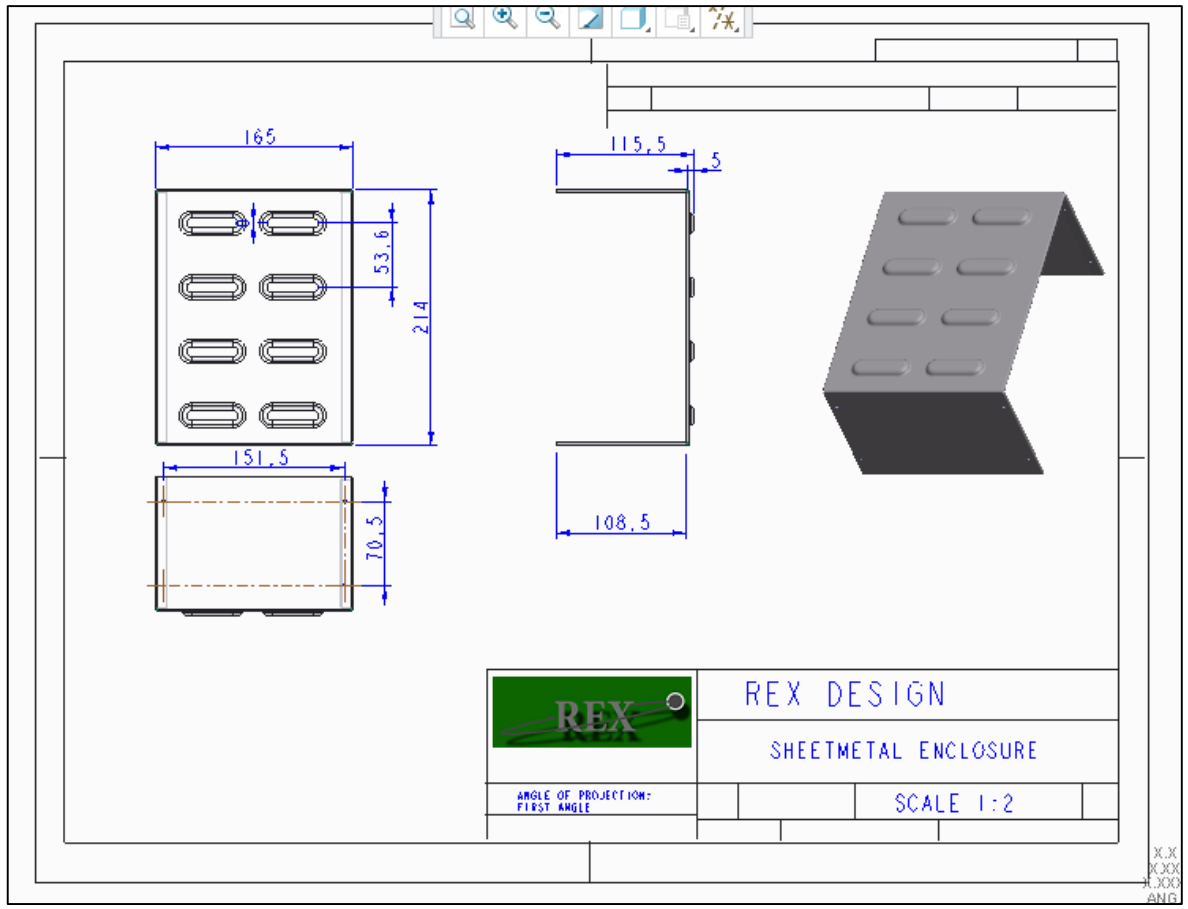


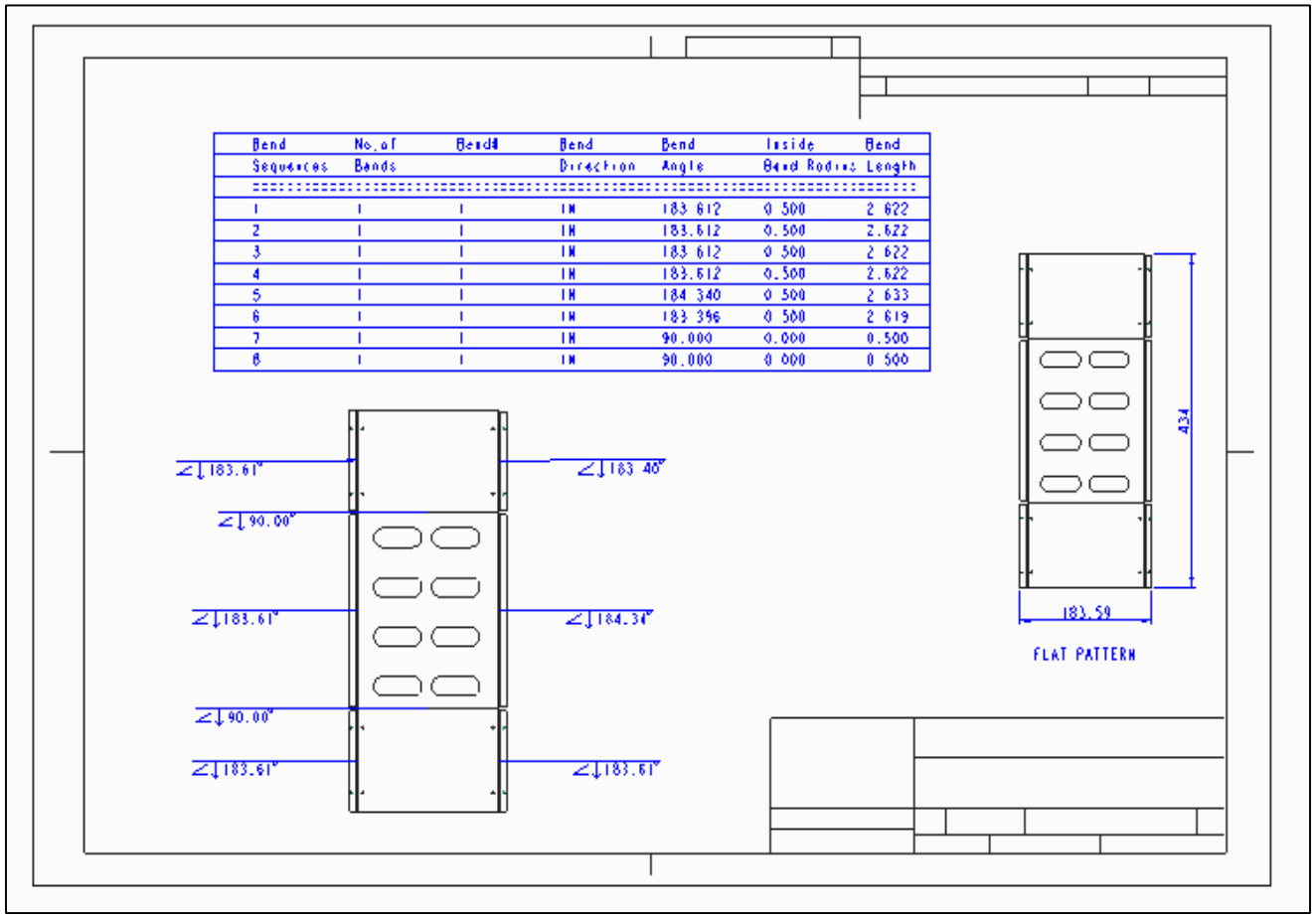

Conducting FEA on your design to find the flaws

# FINAL DESIGN

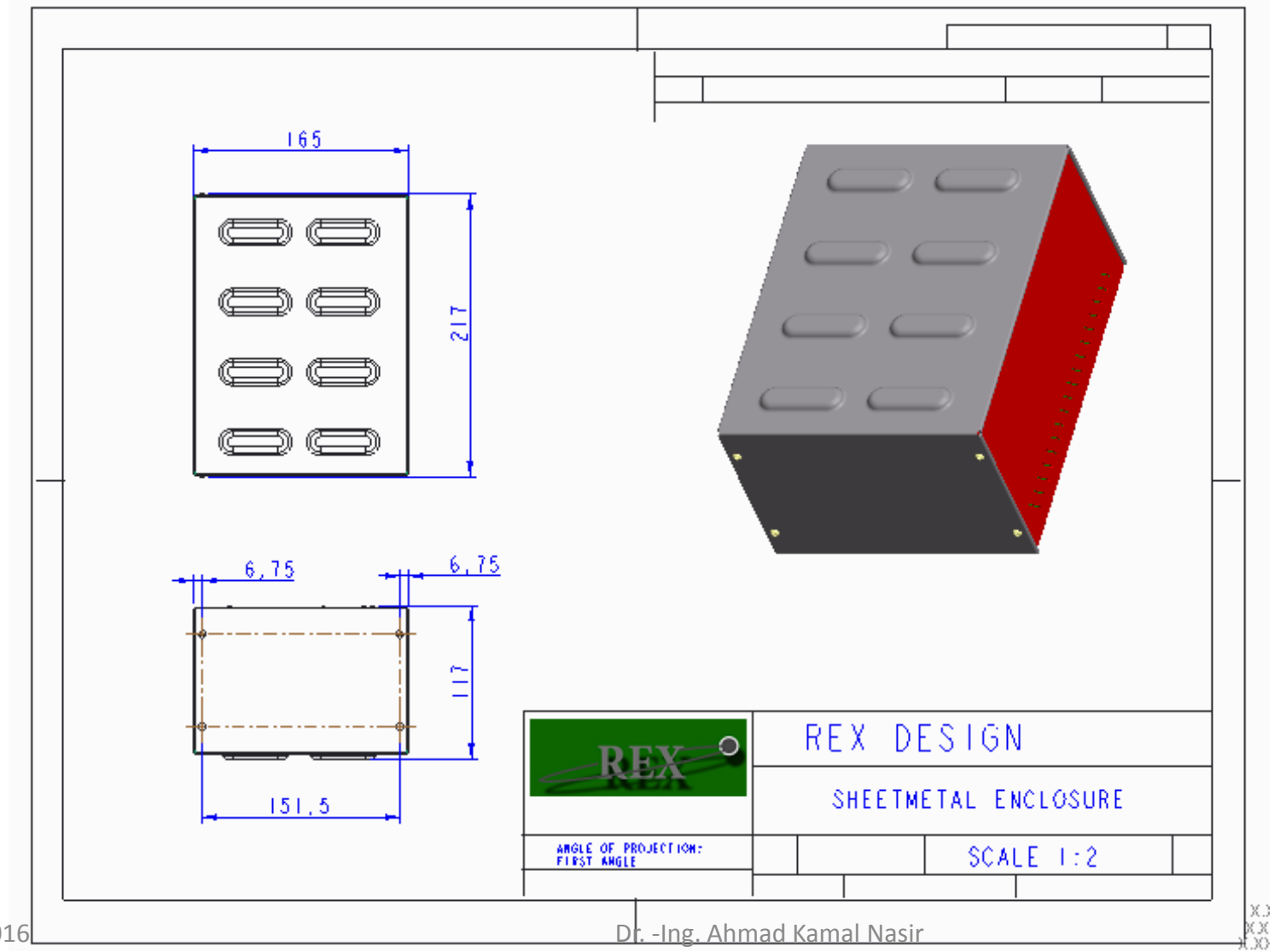

Once the design is frozen, Documentation comes next.

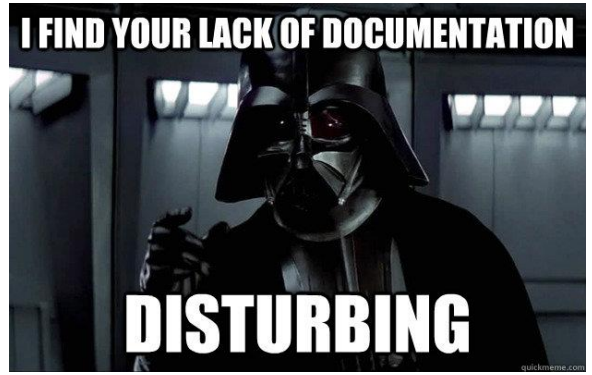
# DESIGN DOCUMENTAION (Drawing and Detailing)

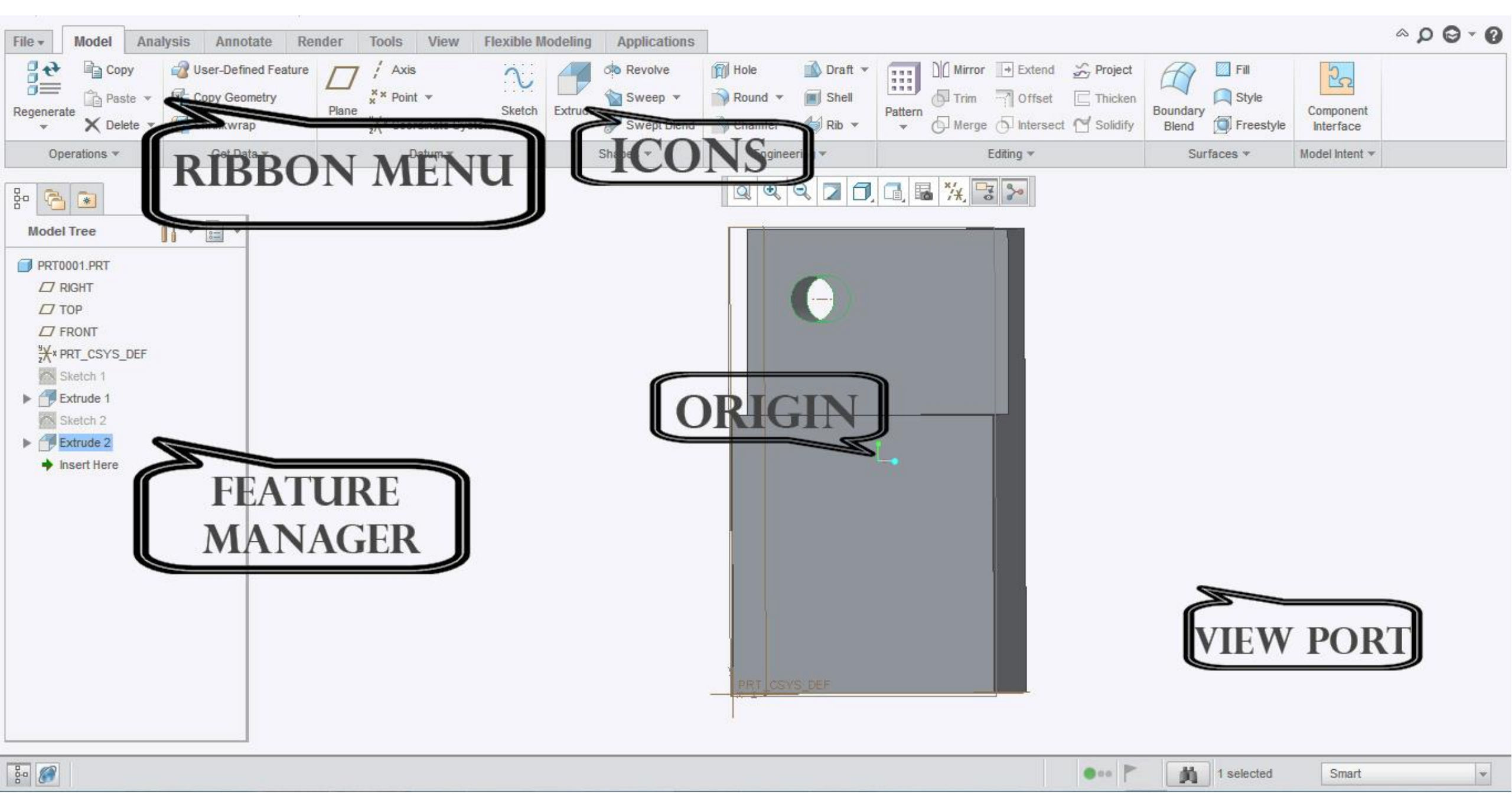


# DESIGN DOCUMENTAION (Drawing and Detailing)



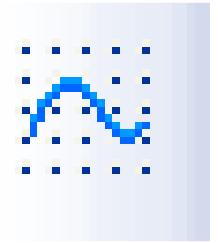




File
Model
Analysis
Annotate
Render
Tools
View
Flexible Modeling
Applications
Regenerate
Copy
Paste
Delete
User-Defined Feature
Copy Geometry
Plane
Axis
Point
Sketch
Extrude
Revolve
Sweep
Swept Blend
Hole
Round
Draft
Shell
Rib
Pattern
Mirror
Trim
Merge
Extend
Offset
Intersect
Project
Thicken
Solidify
Boundary Blend
Fill
Style
Freestyle
Component Interface
Operations
Get Data
Datum
Shapes
Engineering
Editing
Surfaces
Model Intent
RIBBON MENU
ICONS
Model Tree
PRT0001.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Sketch 1
Extrude 1
Sketch 2
Extrude 2
Insert Here
FEATURE MANAGER
ORIGIN
PRT_CSYS_DEF
VIEW PORT
1 selected
Smart

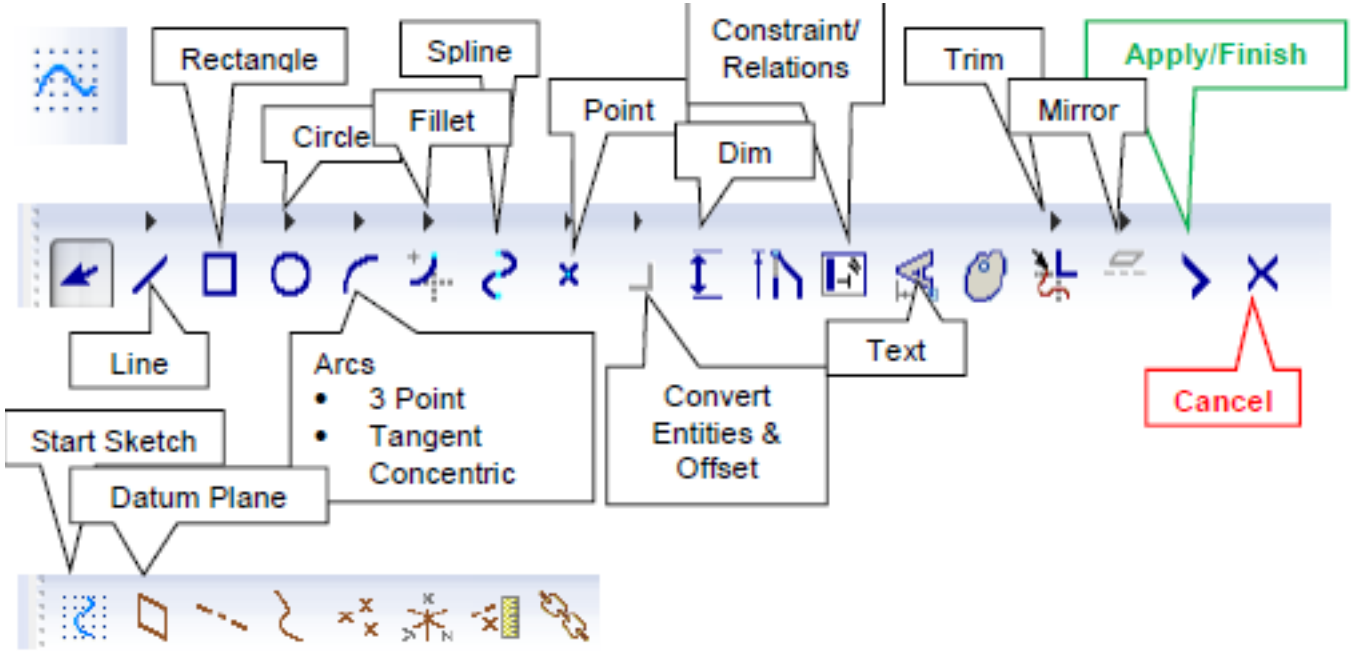
# Creo 2.0 Interface: Mouse Buttons

- **Left Button** - Most commonly used for **selecting** objects on the screen or sketching.
- **Right Button** – Used for activating pop-up **menu** items, typically used when editing. (*Note: you must hold the down button for 2 seconds*)
- **Center Button** – (option) Used for model **rotation**, **dimensioning**, **zoom** when holding Ctrl key, and **pan** when holding Shift key. It also **cancels** commands and line chains.
- **Center Scroll Wheel** – (option) same as Center Button when depressed, only it activates **Zoom** feature when scrolling wheel.

# Sketch

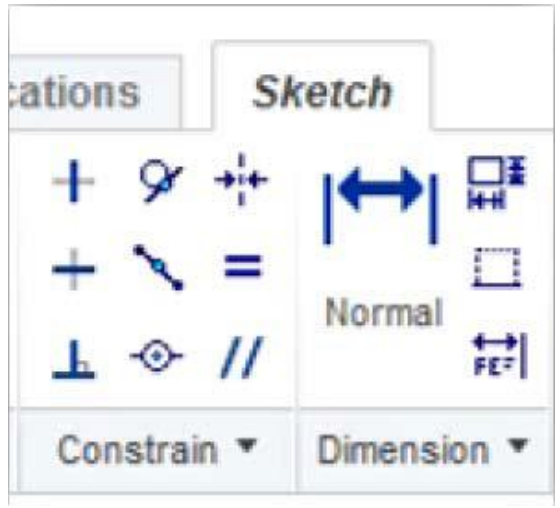


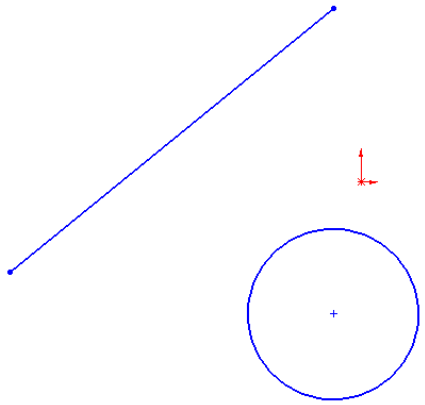
- Typically the first step in any part construction process
- Enables drawing of 2D **datum features**
  - Lines, Rectangles, Circles, Fillet, Arcs etc
- Allows geometry to be controlled using dimensions and constraints
- Sketches can be created on any Datum Plane or Planar Face or Surface

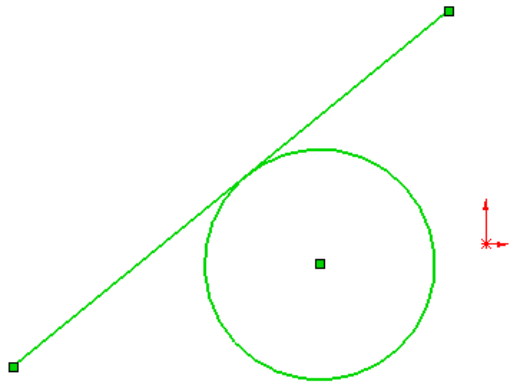


# Controlling Geometry: Constraints

- **Constraints** can be referred to as common elements of geometry such as Tangency, Parallelism, and Concentricity
- These elements can be added to geometric entities automatically or manually during the design process

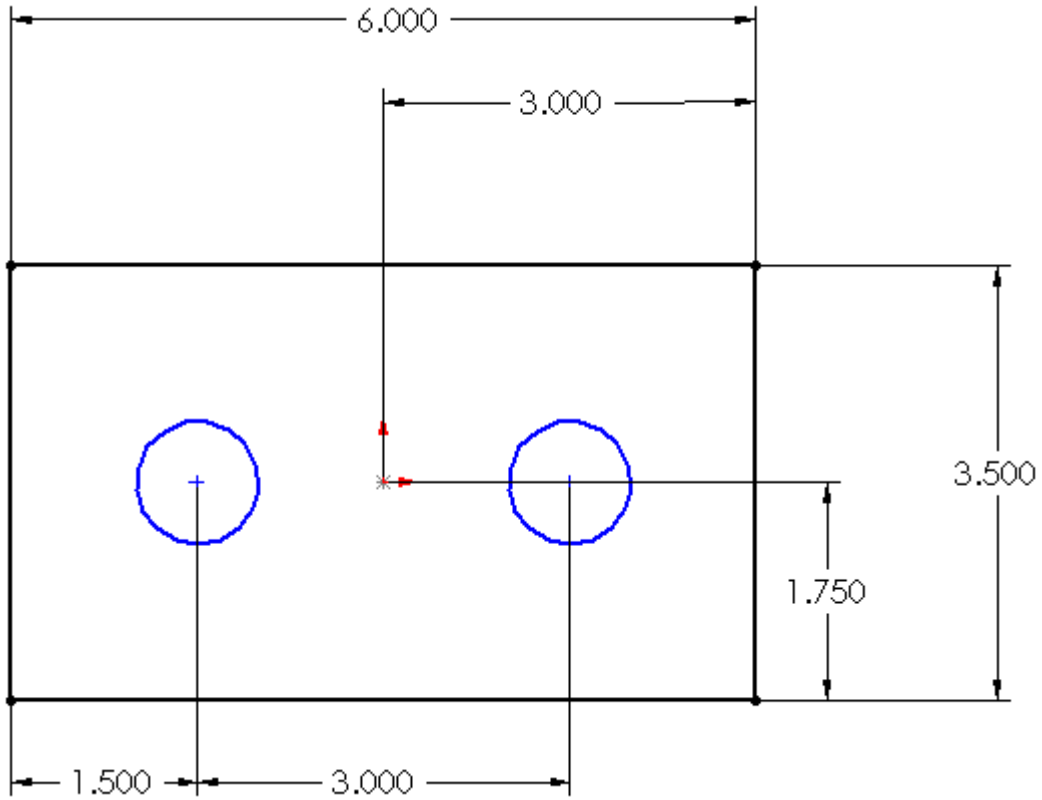


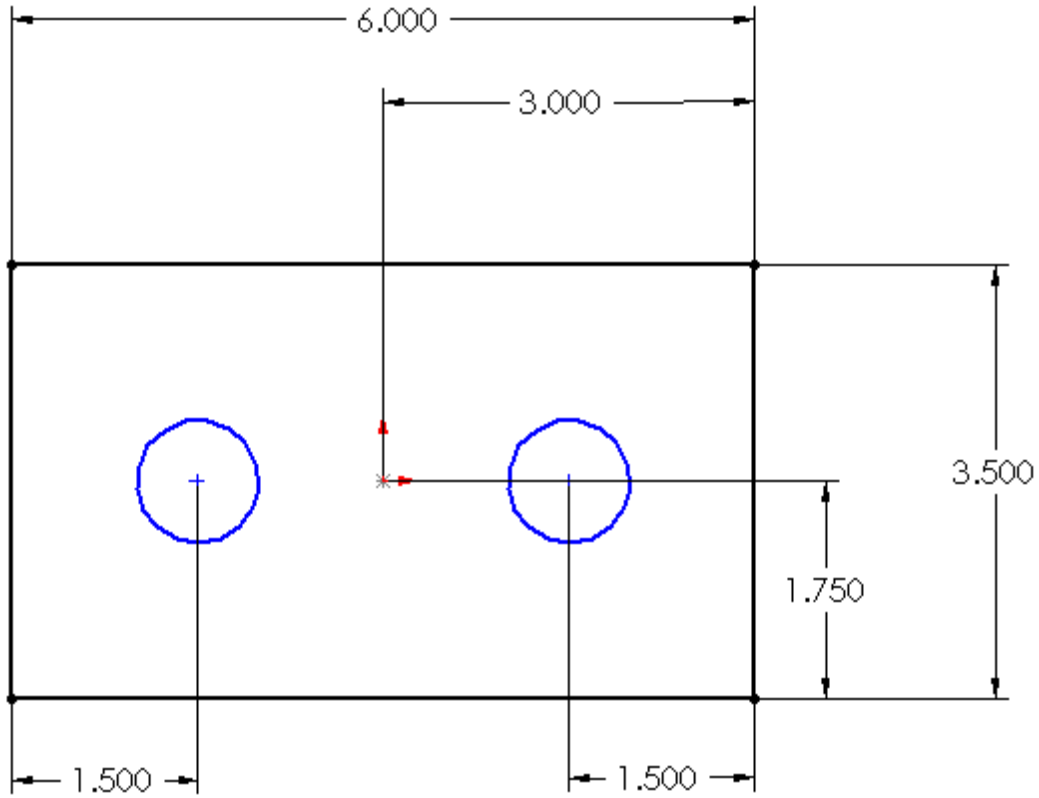


| Constraint | Geometric entities to select | Resulting Constraint |
|---|---|---|
| **Horizontal or Vertical** | One or more lines or two or more points. | The lines become horizontal or vertical (as defined by the current sketch space). Points are aligned horizontally or vertically. |
| **Collinear** | Two or more lines. | The items lie on the same infinite line. |
| **Perpendicular** | Two lines. | The two items are perpendicular to each other. |
| **Parallel** | Two or more lines. A line and a plane (or a planar face) in a 3D sketch. | The items are parallel to each other.<br>The line is parallel to the selected plane. |
| **Tangent** | An arc, ellipse, or spline, and a line or arc. | The two items remain tangent. |
| **Concentric** | Two or more arcs, or a point and an arc. | The arcs share the same centerpoint. |
| **Midpoint** | Two lines or a point and a line. | The point remains at the midpoint of the line. |
| **Coincident** | A point and a line, arc, or ellipse. | The point lies on the line, arc, or ellipse. |
| **Equal** | Two or more lines or two or more arcs. | The line lengths or radii remain equal. |
| **Symmetric** | A centerline and two points,lines, arcs, or ellipses. | The items remain equidistant from the centerline, on a line perpendicular to the centerline. |

# Controlling Geometry: Dimensions




**Holes will always be 3” apart**

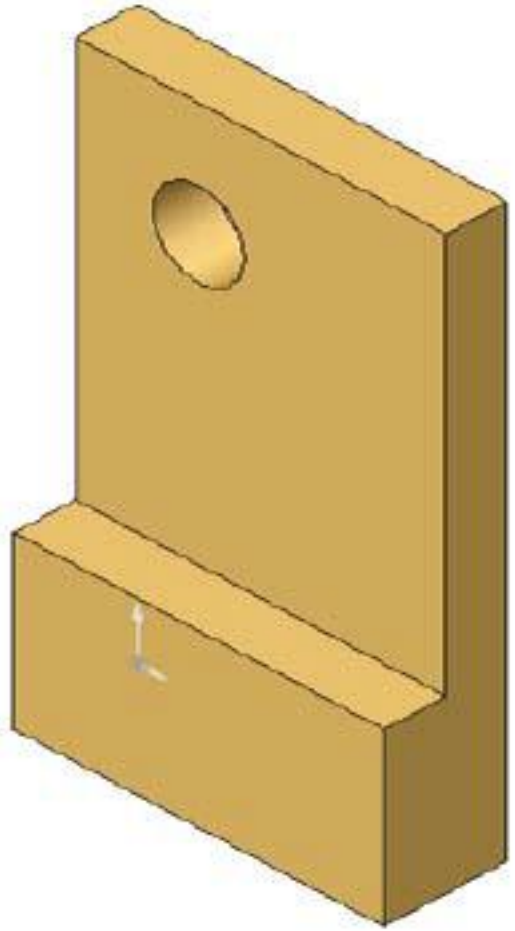



**Holes will always be 1.5” from each edge**

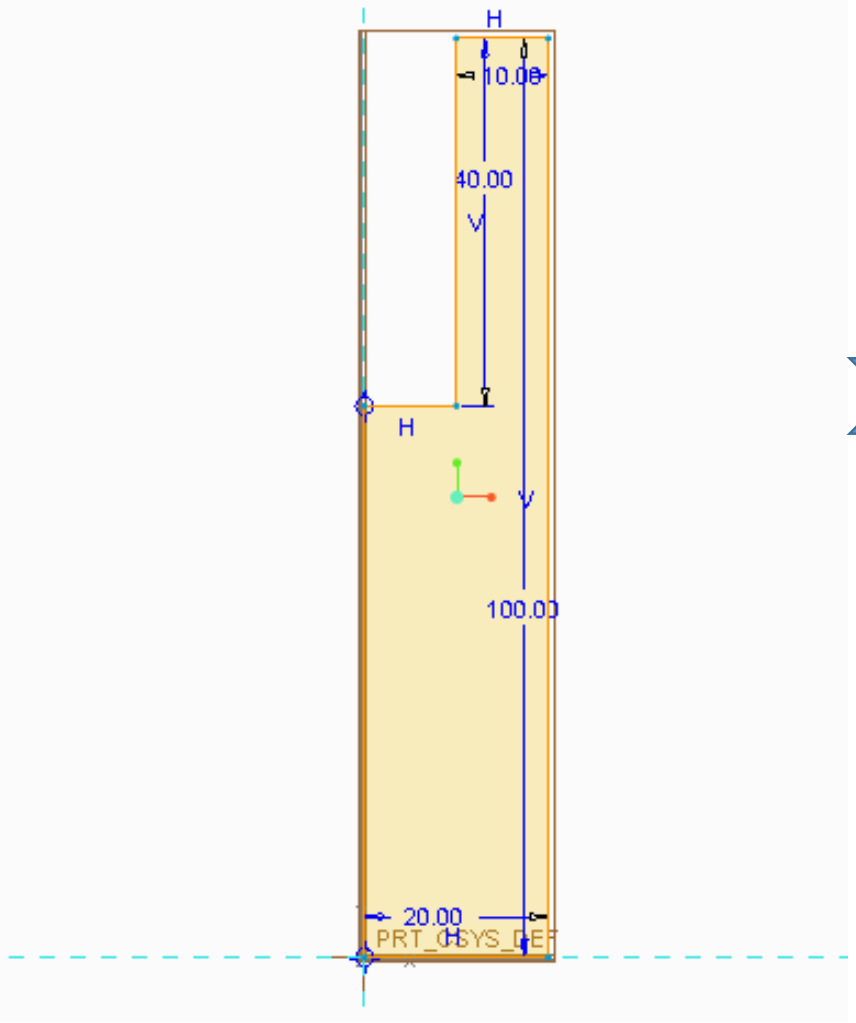
# Demonstration 1

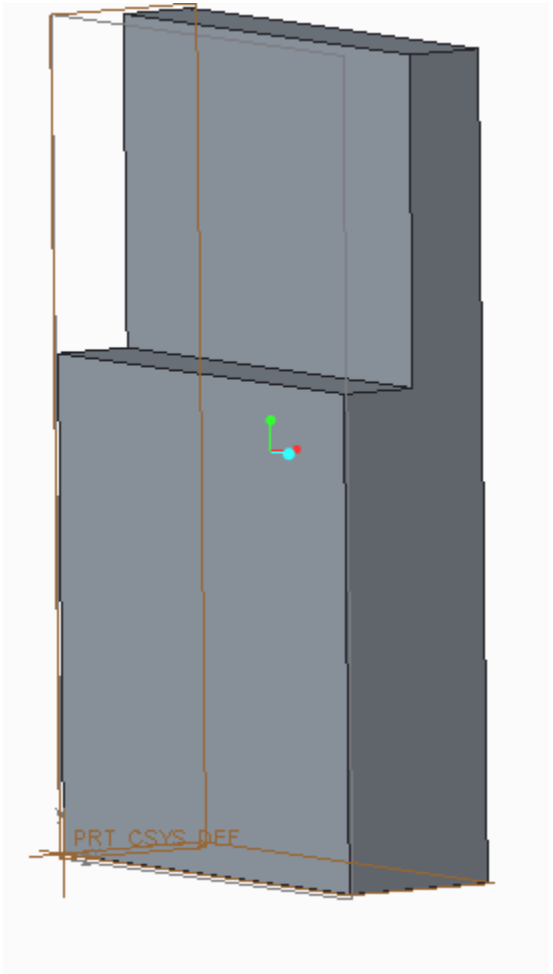
- **Objective:**

Introduction to the Sketch, Extrude, Round, Chamfer and Shell features

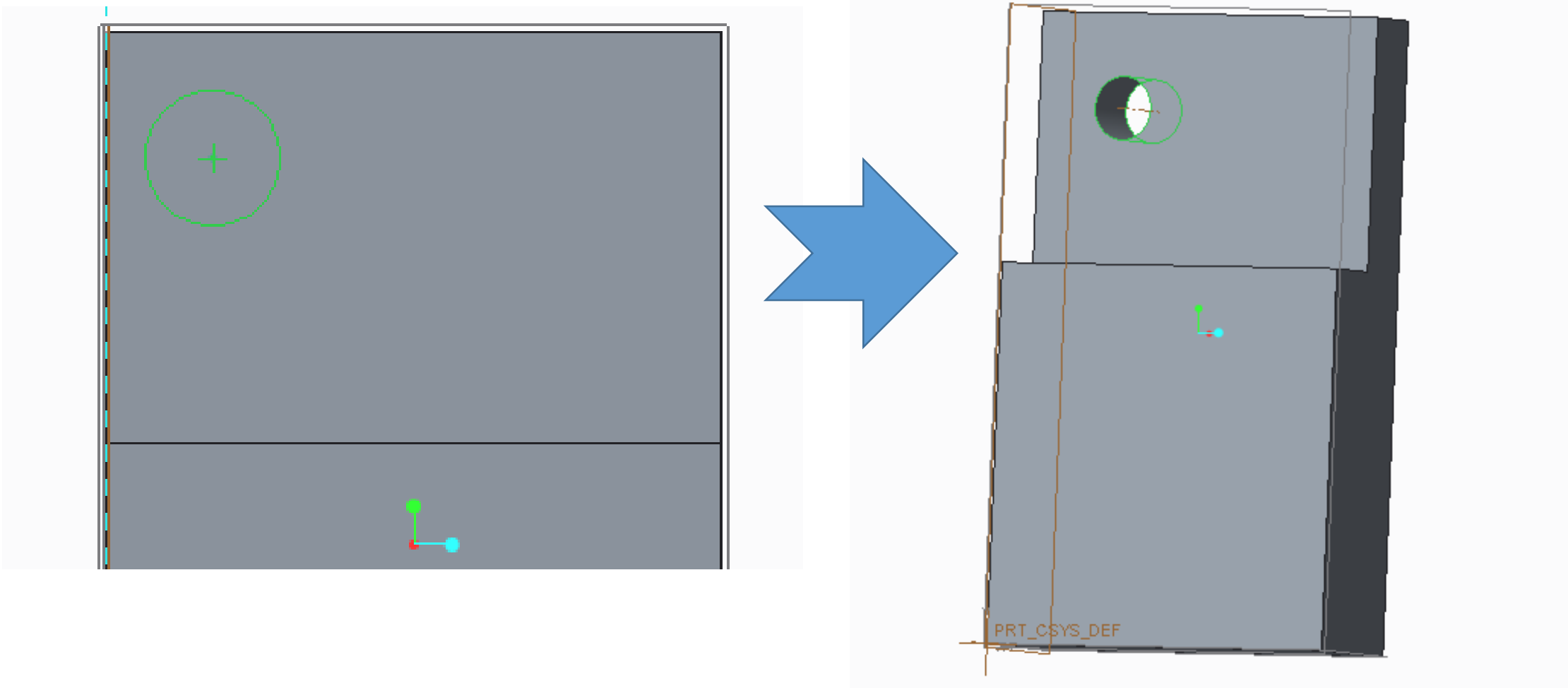


# Demonstration 1 (cont.)

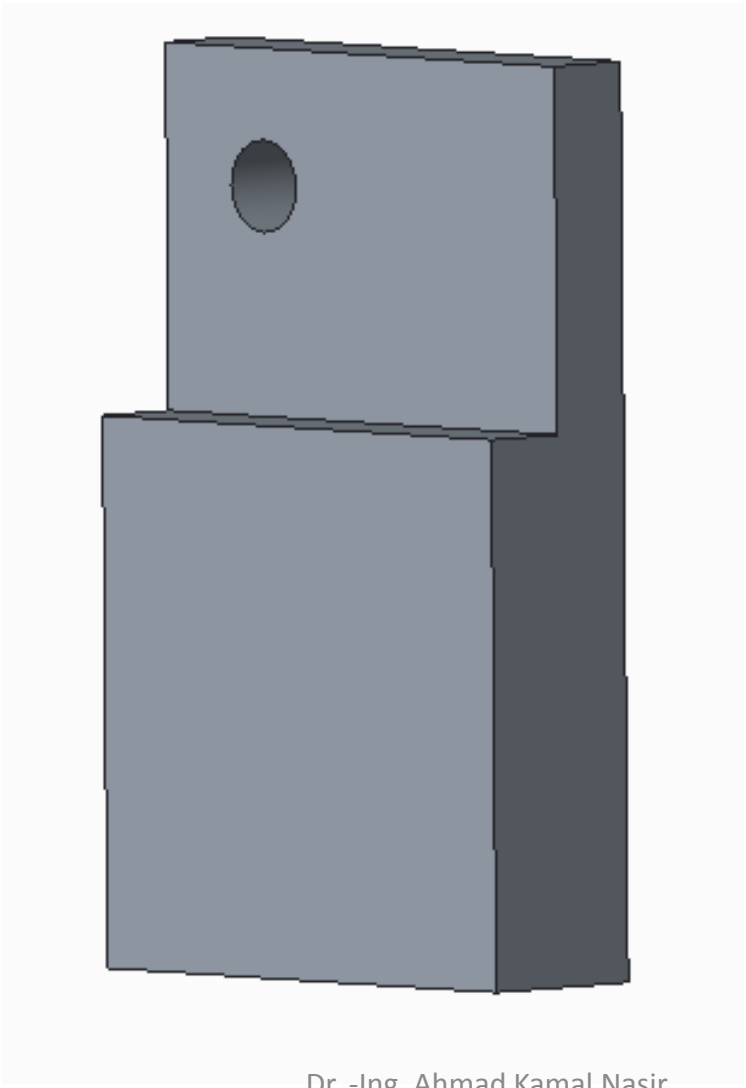


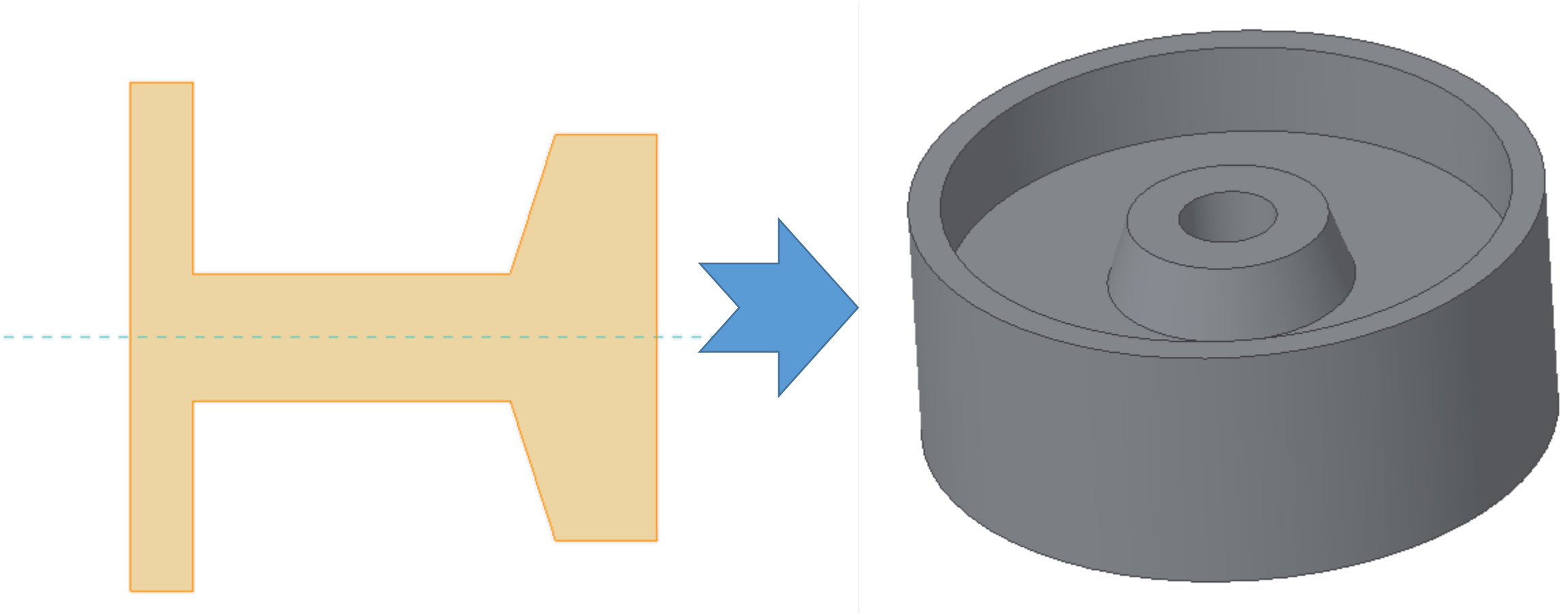
# Demonstration 1 (cont.)



# Demonstration 1 (cont.)



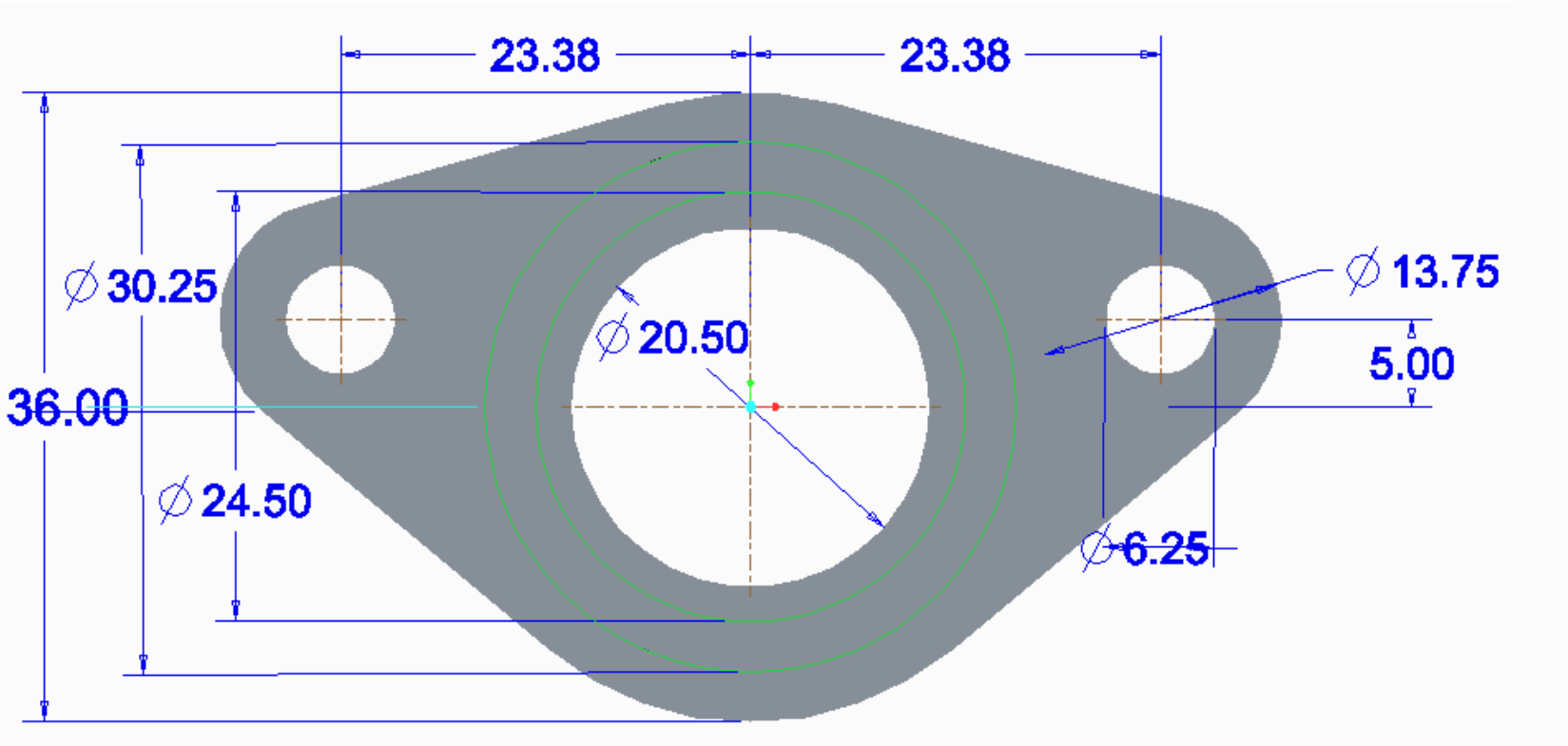
# Demonstration 2

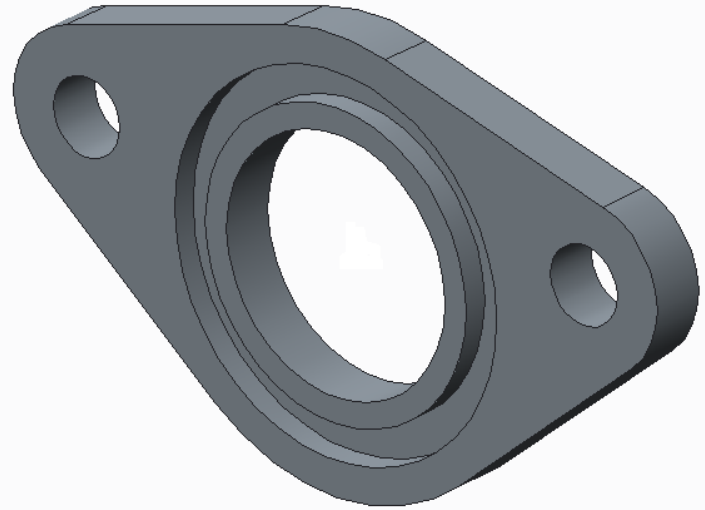


# Summary

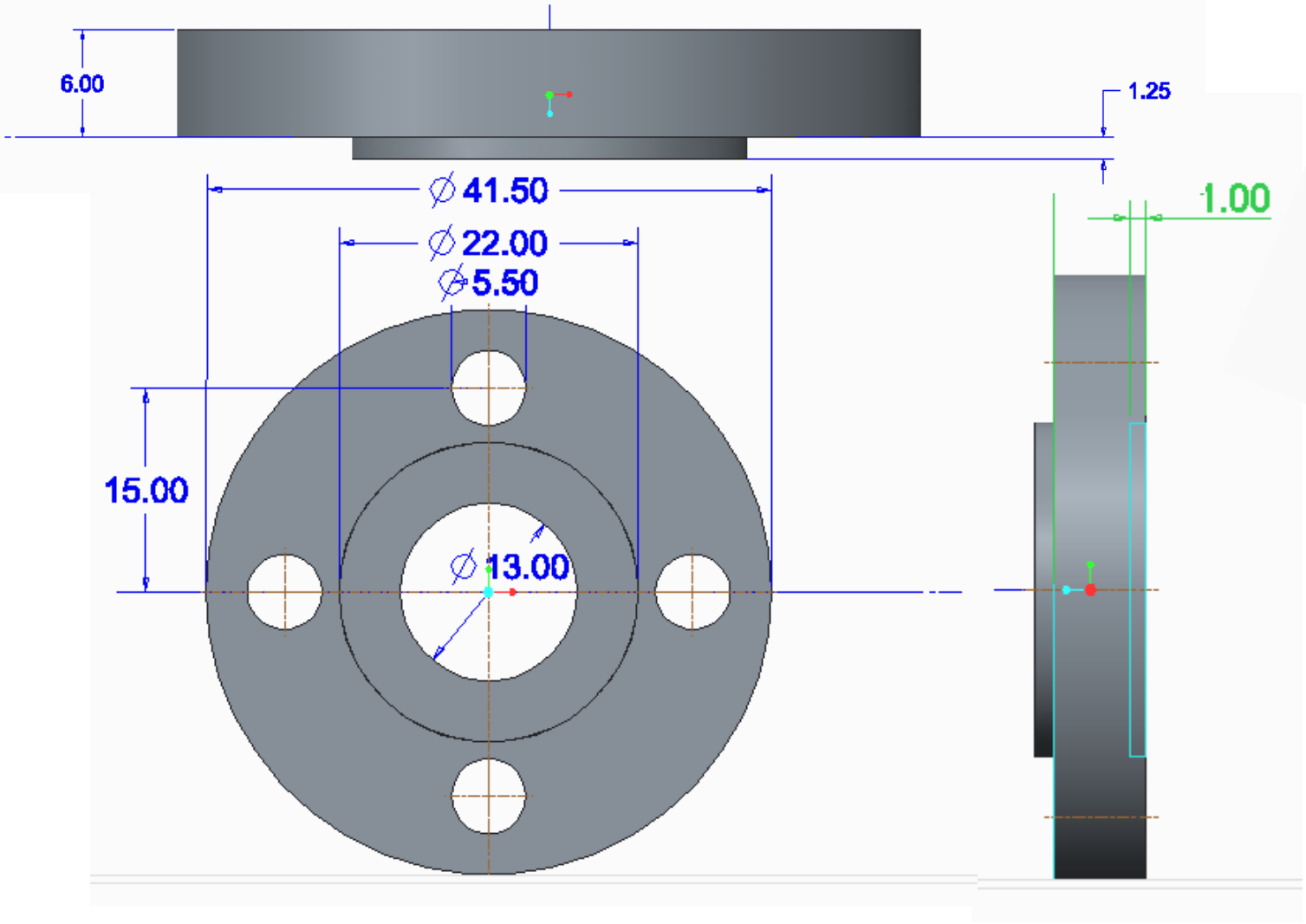
- Computer Aided Modeling
  - Intro to PTC Creo and its features
  - 2D sketching
  - Basics of 3D object modeling
- **Lab Tasks**: 3D part modeling.

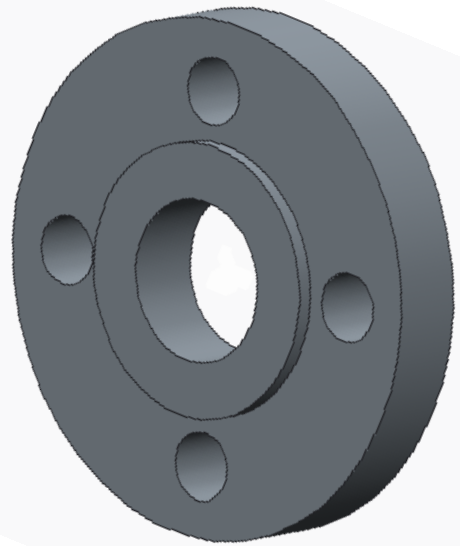
# Lab Task 6(A): Make the part in CREO

# Lab Task 6(B): Make the part in Creo



# Lab Task 6(C): Make the Part in Creo