RTB | RWANDA TVET BOARD

RQF LEVEL 4

GENSD401

COMPUTER SYSTEM AND ARCHITECTURE

SolidWorks Designs

TRAINEE'S MANUAL

October, 2024

RTB | RWANDA TVET BOARD

SOLIDWORKS DESIGN

KOICA Korea International Cooperation Agency

TQUM TVET Quality Management Project

2024

AUTHOR'S NOTE PAGE (COPYRIGHT)

Copies available from:

- *HQs: Rwanda TVET Board-RTB*
- *Web: www.rtb.gov.rw*
- **KIGALI-RWANDA**

Original published version: October 2024

ACKNOWLEDGEMENTS

The publisher would like to thank the following for their assistance in the elaboration of this training manual:

Rwanda TVET Board (RTB) extends its appreciation to all parties who contributed to the development of the trainer’s and trainee’s manuals for the TVET Certificate IV in Computer System and Architecture, specifically for the module **" GENSD401: SolidWorks Designs."**

We extend our gratitude to KOICA Rwanda for its contribution to the development of these training manuals and for its ongoing support of the TVET system in Rwanda.

We extend our gratitude to the TQUM Project for its financial and technical support in the development of these training manuals.

We would also like to acknowledge the valuable contributions of all TVET trainers and industry practitioners in the development of this training manual.

The management of Rwanda TVET Board extends its appreciation to both its staff and the staff of the TQUM Project for their efforts in coordinating these activities.

This training manual was developed:

Under Rwanda TVET Board (RTB) guiding policies and directives

Under Financial and Technical support of

COORDINATION TEAM

RWAMASIRABO Aimable
MARIA Bernadette M. Ramos
MUTIJIMA ASHER Emmanuel

Production Team

Authoring and Review

NSENGIYUMVA Alfred

NZAYISENGA Gad Pacifique

Validation

TUYISHIME Emmanuel

UWAMUKIJIJE R.Donatien

Conception, Adaptation and Editorial works

HATEGEKIMANA Olivier

GANZA Jean Francois Regis

HARELIMANA Wilson

NZABIRINDA Aimable

DUKUZIMANA Therese

NIYONKURU Sylvestre

NIYONZIMA Augustin

Formatting, Graphics, Illustrations, and infographics

YEONWOO Choe
SUA Lim
SAEM Lee
SOYEON Kim
WONYEONG Jeong
NGENDAHAYO Henry Gabriel

Financial and Technical support

KOICA through TQUM Project

TABLE OF CONTENT

ACRONYMS

2D: Two Dimension

3D: Three Dimension

ASM: Assembly

CAD: Computer Aided Design

CADD: Computer Aided Design and Drafting

CAM:Computer Aided Manufacturing

CBT/A: Competency Based Training and Assessment

DRW: Drawing

DWG (.dwg): Drawing

DXF (.dxf): Drawing Exchange Format

HQs: Headquarters

IGES **(.igs, .iges):** Initial Graphics Exchange Specification

KOICA: Korea International Cooperation Agency

PDF (.pdf): Portable Document Format

PRT: Part

RTB: Rwanda TVET Board

SLD: Solid

STEP :Standard for the Exchange of Product

STL (.stl): StereoLithography

SLDPRT: SolidWorks Part File

TQUM Project:TVET Quality Management

TVET: Technical and Vocational Education and Training

TQUM :TVET Quality Management

TVET: Technical and Vocational Education and Training

INTRODUCTION

This trainee's manual includes all the knowledge and skills required in Computer System and Architecture specifically for the module of **"Solidworks Designs.".** Trainees enrolled in this module will engage in practical activities designed to develop and enhance their competencies.

The development of this training manual followed the Competency-Based Training and Assessment (CBT/A) approach, offering ample practical opportunities that mirror real-life situations.

The trainee's manual is organized into Learning Outcomes, which is broken down into indicative content that includes both theoretical and practical activities. It provides detailed information on the key competencies required for each learning outcome, along with the objectives to be achieved.

As a trainee, you will start by addressing questions related to the activities, which are designed to foster critical thinking and guide you towards practical applications in the labor market. The manual also provides essential information, including learning hours, required materials, and key tasks to complete throughout the learning process.

All activities included in this training manual are designed to facilitate both individual and group work. After completing the activities, you will conduct a formative assessment, referred to as the end learning outcome assessment. Ensure that you thoroughly review the key readings and the Points to Remember section.

MODULE CODE AND TITLE: GENSD401 SOLIDWORKS DESIGNS

Learning Outcome 1: Set up SolidWorks space
Learning Outcome 2: Create Part
Learning Outcome 3: Assemble parts
Learning Outcome 4: Produce drawing

Learning Outcome 1: Set up Solidworks Space

Indicative contents

1.1.Installation of SolidWorks software

1.2.Selection of SolidWorks template

1.3. Interface customization

Key Competencies for Learning Outcome 1: Set Up SolidWorks space

Knowledge	Skills	Attitudes
• Definition of CAD software • Description of SolidWorks software • Description of solid work templates	• Selecting suitable SolidWorks set up • Installing SolidWorks software • Launching SolidWorks software • Demonstrating customer user Interfaces • Customizing SolidWork user interface	• Taking initiative • Being Problem Solver • Being Critical thinker • Having Passion • Having Adaptability • Being Decision Maker • Being Team worker • Having Creativity

Duration: 10 hrs

Learning outcome 1 objectives:

By the end of the learning outcome, trainee will be able to:

1. Define clearly Computer Aided Design(CAD) as used in Computer System and Architecture.
2. Describe clearly SolidWorks software as used in Computer System and Architecture.
3. Install correctly SolidWorks software based on computer system requirements.
4. Describe properly SolidWorks Template /document based on its function.
5. Select properly SolidWorks template according to the task requirements.
6. Customize properly SolidWorks software user interfaces refer to the assigned tasks.

Resources		
Equipment	**Tools**	**Materials**
• Computer • Projector	• Storage media devices • SolidWorks Software full package	• electricity • Internet connectivity

Indicative content 1.1: Installation of SolidWorks Software

Duration: 4 hrs

Theoretical Activity 1.1.1: Description of SolidWorks Software

Tasks:

1: Answer the following questions:

i. Define the following key terms: CAD, CAM, CADD, CAE, 2Dmodeling and 3Dmodeling?

ii. Describe what is SolidWorks software?

iii. What are the applications of SolidWorks software?

iv. List advantages and limitations of SolidWorks software.

v. Highlight computer system requirements before installing SolidWorks?

2: Provide your answers on paper/flipchart.

3: Present your findings.

4: Pay attention on trainer's clarifications and ask questions if any.

5: For more clarification, read the key readings 1.1.1 in trainee manual

Key readings 1.1.1.: Description of SolidWorks software

- **Definitions of Terms:**
  - ✔ **CAD (computer-aided design):** It is the use of computer-based software to aid in design processes. CAD software is frequently used by different types of engineers and designers. CAD software can be used to create two-dimensional (2D) drawings or three-dimension(3D) models
  - ✔ **CAM (Computer-Aided Manufacturing):** CAM involves using computer software and machinery to control and automate manufacturing processes.
  - ✔ **CADD (Computer-Aided Design and Drafting):** CADD is a term used to describe the combination of CAD and drafting, which involves creating technical drawings with precise dimensions and annotations.
  - ✔ **CAE (Computer-Aided Engineering):** CAE involves using computer software to simulate and analyze a status of designs under different conditions. It helps engineers to evaluate factors such as stress, thermal effects.

- ✔ **2DModeling:** 2Dmodeling refers to the creation of digital representations of objects or designs in two dimensions, typically using lines, curves, and shapes. It involves working with shapes and drawings in a two-dimensional space, length and width, without depth or perspective.
- ✔ **3DModeling:** 3D modeling involves creating three-dimensional digital representations of objects or designs. It allows for the creation of complex and realistic models that can be viewed and analysed from different angles. refers to the three spatial dimensions of width, height and depth SolidWorks description.
- ✔ **SolidWorks** is a 3D parametric design software that is used to design all sorts of products such as automobiles, computer peripherals, marine equipment, airplane parts, cell phones, cameras, furniture, electrical assemblies, glasses, lighting fixtures, toys, vacuum cleaners, or any other like products. it was developed by Massachusetts Institute of Technology (MIT) graduate Jon Hirsch tick and was bought by Dassault Systems in 1997. The software now encompasses a number of programs that can be used for both 2D and 3D design.

- **Application of SolidWorks**

✔ Some common applications of SolidWorks include:

- Mechanical Engineering.
- Product Design and Development.
- Architecture and Construction.
- Electrical and Electronics.
- Automotive and Transportation.
- Aerospace and Defense.

- **Advantages and limitations of SolidWorks**

✔ **Advantages of SolidWorks:**

- User-Friendly Interface: SolidWorks provides an intuitive and user-friendly interface that makes it easy to learn and use for both beginners and experienced users.
- Comprehensive Design Tools: SolidWorks offers a wide range of tools and features for creating complex 3D models, performing simulations, generating technical drawings, and collaborating with others.
- Parametric Modeling: SolidWorks supports parametric modeling, allowing users to easily make changes to designs and have those changes automatically update throughout the model.
- Integration with Other Software: SolidWorks can integrate with other software applications, such as simulation software, data management systems, and manufacturing software, facilitating a seamless workflow.
- Extensive Online Community and Support: SolidWorks has a large online community of users and resources, including forums, tutorials, and documentation, providing support and assistance for users.

✔ **Limitations of SolidWorks:**

- Cost: SolidWorks is a commercial software and can be expensive, especially for individual users or small businesses.
- System Requirements: SolidWorks requires a computer system with sufficient hardware resources to run smoothly. Older or lower-spec systems may struggle to handle complex designs.
- Steep Learning Curve: While SolidWorks has a user-friendly interface, learning and mastering all the advanced features and capabilities may require time and effort.
- Software Updates: SolidWorks releases new versions and updates regularly, which may require users to adapt to changes in the software and may require additional training or support.

- **Computer system requirements**

  The specific requirements may vary slightly depending on the version of SolidWorks you're installing. It's always recommended to check the official SolidWorks documentation for the most up-to-date information.

  ✔ **System Requirements:**

  - Operating System: Windows 11 or Windows 10 (64-bit)
  - Processor: Intel® Core i5, i7, or equivalent AMD® processor
  - Memory: 8GB or more
  - Hard Drive: Standard or Solid State Drive (SSD) with at least 250GB free space

Practical Activity1.1.2: Installing SolidWorks software

Task:

1. You are asked to perform the following task:

   As a computer technician, you are requested to go to the computer lab to install SolidWorks software on the computers.
2. Read the steps from key readings 1.1.2 in trainee manual
3. Follow the demonstration from your trainer of how to install SolidWorks.
4. Install SolidWorks software by simulating the demonstration of trainer.
5. Ask for support if there is an occurred issue.

Key readings 1.1.2: Installing SolidWorks software

Installing SolidWorks software Steps :

Step 1.Ensure that the internet is disconnected to computer

Step 2.Turn off any antivirus computer window protection and firewalls

Step 3. Go to control panel then click on programs and features for turning window features on/off

Step 4. Ensure.netframework3.5(includes Net2.0and 3.0) and ensure.netframework4.8 advanced services

Step 5. Go to downloaded SolidWorks folder then unzip that folder and unzip crack folder

Step 6. After Do double clicking on serial licensing then click yes to ensure that all serial licensing is added to registry

Step 7. Click on SolidWorks crack folder then copy flex.net_server

Step 8. Go to local disc and paste it in local disc C

Step 9. Click on flex.net server install and run as administrator

Step 10. After you will go back to downloaded unzipped SolidWorks folder then do double click on SolidWorks set up.exe

Step 11. click ok

Step 12. click next

Step 13. Click install now then click ok

Step 14. Click finish

Step 15. Don’t open SolidWorks until you followed these steps for activation or cracking

Step 16. Do double clicking SolidWorks enabler code for adding code to registry

Step 17. Go to crack folder and copy SOLIDWORKS CORP into program file in order to replace the existing

Step 18. Open SolidWorks and start navigating

Points to Remember

Term definition:

- CAD (Computer-Aided Design) uses software for design processes, enabling the creation of 2D drawings and 3Dmodels.
- CAM (Computer-Aided Manufacturing) automates manufacturing processes through software and machinery.
- CADD (Computer-Aided Design and Drafting) combines CAD with drafting to produce technical drawings with precise dimensions.
- CAE (Computer-Aided Engineering) simulates and analyzes designs under various conditions, assessing factors like stress and thermal effects.
- 2D vs. 3DModeling: 2Dmodeling creates flat representations, while 3Dmodeling produces complex digital objects viewable from multiple angles.

✔ **SolidWorks Overview:**

SolidWorks is a 3D parametric design software used for various industries, developed by Jon Hirsch tick and acquired by Dassault Systems in 1997.

- ✔ **Applications of SolidWorks:** Common uses of SolidWorks include mechanical engineering, product design, architecture, automotive, aerospace, and electronics.
- ✔ **Advantages:** SolidWorks features a user-friendly interface, comprehensive design tools, parametric modeling, integration capabilities, and strong community support.
- ✔ **Limitations:** Challenges include high costs, hardware requirements, a steep learning curve, and the need for regular software updates.
- ✔ **System Requirements:**
- ✔ Recommended specifications include Windows 10/11 (64-bit), Intel Core i5/i7 or equivalent AMD processor, 8GB RAM, and 250GB free storage space.
- ✔ **Installing SolidWorks software Steps:**

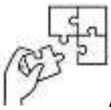
1) Disconnect Internet: Ensure the computer is offline before starting the installation.
2) Disable Security Software: Turn off any antivirus programs and firewall protection.
3) Check Windows Features: Go to Control Panel, then Programs and Features, to enable necessary Windows features.
4) Verify .NET Framework: Ensure .NET Framework 3.5 (including 2.0 and 3.0) and .NET Framework 4.8 Advanced Services are installed.
5) Prepare Installation Files: Navigate to the downloaded SolidWorks folder, unzip it, and also unzip the crack folder.
6) Add Serial Licensing: Double-click the serial licensing file and confirm to add it to the registry.
7) Install Flex.Net Server: Copy the Flex.Net Server file to the C: drive and run the installer as an administrator.

8) Run SolidWorks Setup: In the unzipped SolidWorks folder, double-click on setup.exe to begin the installation.

9) Follow Installation Prompts: Click "OK," then "Next," and select "Install Now," followed by "Finish."

10) Activation Steps: Do not open SolidWorks until you complete the activation steps, including adding the SolidWorks enabler code to the registry and replacing files in the program directory.

11) Start Using SolidWorks: Once all steps are completed, open SolidWorks and begin navigating.



Application of learning 1.2.

ABD computer peripheral manufacturing company ltd, wants to hire a technician who is able to install SolidWorks software because this company has a problem of not having enough computers for satisfying their clients. As a technician you are hired to install SolidWorks software to their computers. A task includes ensuring that the installation process is smooth, software is fully operational and any potential installation issues are addressed effectively.

Indicative content 1.2: Selection of SolidWorks template

Duration: 3hrs

Theoretical Activity 1.2.1: Description of SolidWorks Template

Tasks:

1. You are asked to answer the following questions:
   i. Describe the three primary types of SolidWorks templates/documents?
   ii. What are primarily the SolidWorks file extension formats?
   iii. What are the most widely used drawing template standards?
   iv. What are Some of the most widely used drawing template standards?
2. Provide the answer for the questions and write them on papers/ flipchart.
3. Present the findings/answers
4. Pay attention on trainer's clarifications and ask questions if there are any.
5. For more clarification, read the key readings 1.2.1

Key readings 1.2.1.: Description of SolidWorks Template

There are three primary types of SolidWorks documents (template):

1. **Part template /Document (. sldprt)**

- ✔ It is used to create individual 3D components.
- ✔ It contains the geometry and design details of a single part. You can create solid models, sheet metal parts, weldments, and more using various sketching, extrusion, and feature tools.
- ✔ Designing individual components that will later be assembled together in an assembly document.
- **File Extension:** sldprt

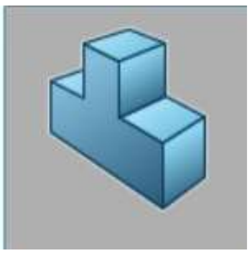


Image 1:Part Template

2. **Assembly template /Document (.sldasm)**

- ✔ It is Used to assemble multiple parts into a single, functional unit.

- ✔ It allows the user to position and mate parts together to define the relationships between components. Assemblies can also include sub-assemblies (assemblies within assemblies).
- ✔ Creating complex products or systems by combining individual parts, such as mechanical devices, computer peripherals, machinery, and consumer products.

Image2: Assembly Template

3. Drawing Template /Document (.slddrw)

- ✔ It is used to create 2D representations of parts or assemblies.
- ✔ It generates detailed drawings from parts or assemblies, including views (front, side, top), dimensions and annotations. These drawings are essential for manufacturing and documentation purposes.
- ✔ Providing detailed, standardized blueprints for manufacturing, fabrication, or assembly processes.

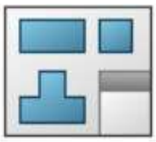


Image 3: Drawing Template

- **Some of the most widely used drawing template standards:**
- ✔ ISO (International Organization for Standardization): Offers a comprehensive set of standards for various industries, including mechanical engineering, construction, and manufacturing.
- ✔ ANSI (American National Standards Institute): Provides standards primarily used in the United States, often aligned with ISO standards.
- ✔ ASME (American Society of Mechanical Engineers): Focuses on standards for mechanical engineering, including design, manufacturing, and testing.
- ✔ JIS (Japanese Industrial Standards): Used primarily in Japan and is often compatible with ISO standards.
- ✔ DIN (German Institute for Standardization): Widely used in Germany and Europe, offering a range of standards for various industries.
- **Note that:** The paper size used in SolidWorks depends on the specific ISO standard you are following and the type of drawing you are creating.

Practical Activity 1.2.2: Selecting SolidWorks Template

Tasks:

1: Read and answer the following question:

As computer technician, you are asked to select at least one SolidWorks Template and read its interfaces.

2: Provide the answer for the asked question and write them on papers.

3: Present the findings/answers to the whole class

4: For more clarification, read the key **readings 1.2.1.**

5: In addition, ask questions where necessary.

Key readings 1.2.1 : Selecting SolidWorks Template

1. Part Template.

Steps:

1. Create a new part: Go to File > New > Part.
2. Define the template's properties: Set the units and document name
3. Add features and dimensions: Create the basic geometry or features that you want to include in your template.
4. Set material and other properties: Assign the desired material and define other properties as needed.
5. Save the template: Go to File > Save As and choose a location to save the template. Give it a descriptive name with a .sldprt extension.

2.Assembly Template

Steps:

1. Create a new assembly: Go to File > New > Assembly.
2. Define the template's properties: Set the units, document name, and other relevant properties.
3. Insert components: Add the components that you want to include in your template.
4. Define relationships and constraints: Establish the desired relationships and constraints between the components.

5. Save the template: Go to File > Save As and choose a location to save the template. Give it a descriptive name with a .sldasm extension.

3.Drawing Template

Steps:

1. Create a new drawing: Go to File > New Drawing.

2. Define the template's properties: Set the units and sheet size

3. Insert views: Add the desired views of the part or assembly to the drawing.

4. Add dimensions and annotations: Create dimensions and notes

5. Save the template: Go to File > Save As and choose a location to save the template. Give it a descriptive name with a .slddrw extension.

Points to Remember

- There are three primary types of SolidWorks documents **(template):**
  - ✔ **Part template /Document (.sldprt)**
  - ✔ **ASSEMBLY TEMPLATE /Document (.sldasm)**
  - ✔ **Drawing Template /Document (.slddrw)**
- **SolidWorks primarily uses the following file extension formats:**
  - **sldprt**
  - **sldasm**
  - **slddrw**
- **Some of the most widely used drawing template standards:**
  - **ISO**
  - **ANSI**
  - **ASME**
  - **JIS**
  - **DIN**
- In selecting SolidWorks Templates trainee must know the steps of Selecting SolidWorks Templates

Application of learning 1.2.

ABD computer peripheral manufacturing company ltd, wants to hire a technician who is a professional worker of CAD Software such as SolidWorks, As a technician first turn on computer, launch SolidWorks Software and Select part, Assembly and drawing template.

Indicative content 1.3: Interface customization

Duration: 3 hrs

Theoretical Activity 1.3.1: Description of SolidWorks customer Interfaces

Tasks:

1: Answer the following questions refer to the introduction:

- **i.** Identify SolidWorks software Interfaces
- **ii.** What is the purpose of customizing Toolbars and Menus?
- **iii.** Identify the functions of tool bars and menu bar customization
- **iv.** What is the purpose of customizing Keyboard shortcuts?
- **v.** What are the functions of using keyboard shortcuts in SolidWorks?
- **vi.** What are the functions of customizing the Command Manager in SolidWorks?
- **vii.** What is the purpose of customizing the Command Manager in SolidWorks?
- **viii.** What are the functions of customizing the Custom Colours and Themes ?
- **ix.** Highlight the functions of mouse gestures
- **x.** What are the functionsof customizing the Custom Property Tab Builder ?

2: Write the findings on paper or flip chart

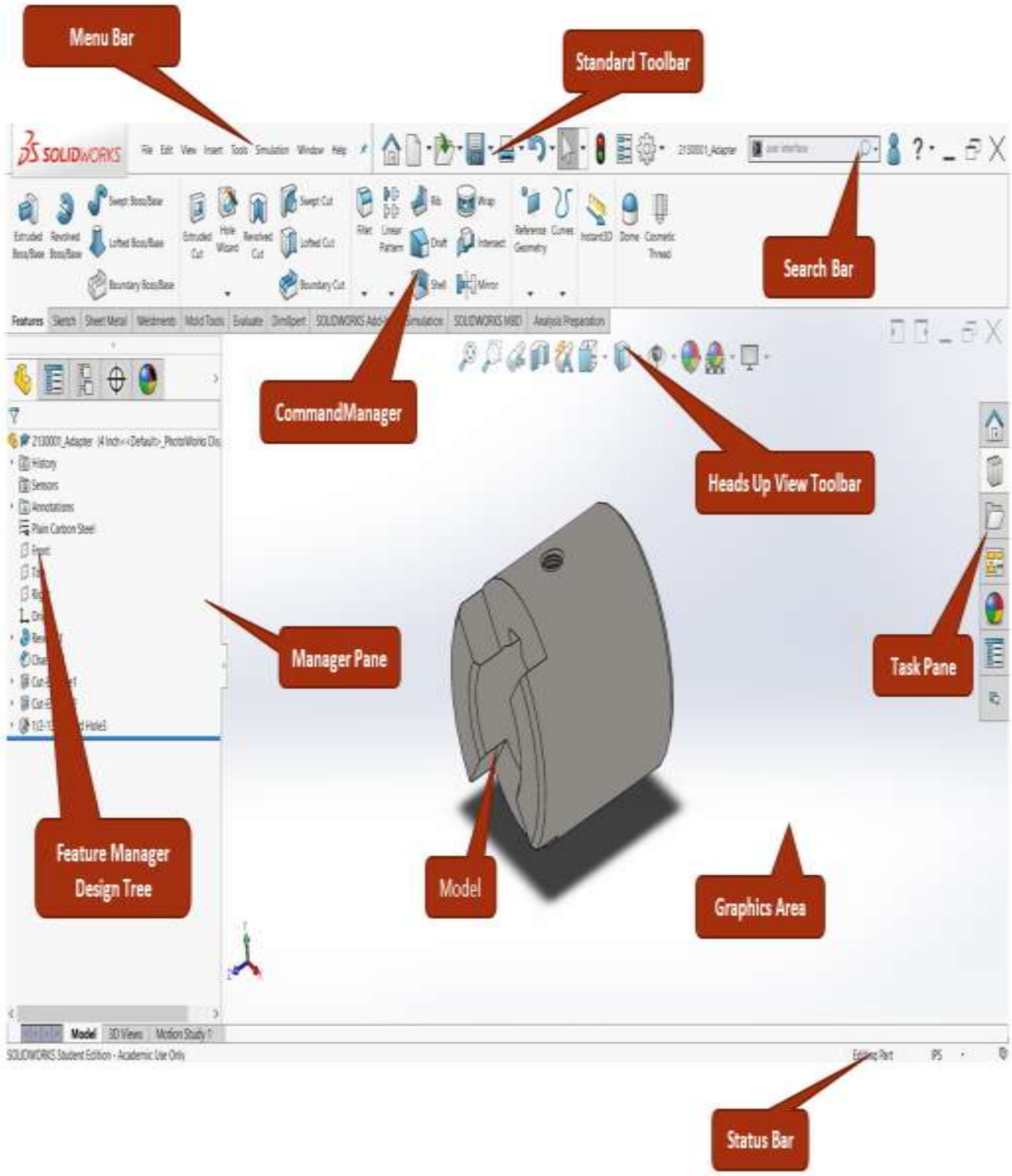
3: Present the findings/answers on paper/flip chart

4: Pay attention on trainer’s clarifications and ask questions if any.

5: Read the key readings 1.2.1 for more clarification

Key readings 1.3.1.: Description of SolidWorks customer Interfaces

- **The SolidWorks user Interface is primarily comprised of nine major sections:**
  1. The Menu Bar
  2. The SOLIDWORKS Menus
  3. The Quick Access Tools
  4. The Command Manager
  5. The Feature Manager Design Tree
  6. The Heads-Up View Toolbar
  7. The Graphics Area
  8. The Task Pane
  9. The Status Bar



- **Interface Customization**

✔ **Toolbars and Menus Customization:** Organize frequently used commands and features for easy access.

- **Toolbars:** Horizontal or vertical bars that contain icons or buttons representing frequently used commands. You can customize their appearance, content, and arrangement to suit your preferences.
- **Function of tools bar Customization:**

1. Create custom toolbars with specific commands.
2. Rearrange icons within existing toolbars.

- **Menus:** Hierarchical lists of commands that can be accessed by clicking on a menu bar or a button. You can customize the items within menus, their arrangement, and even create new menus
- **Function of menu bar Customization:**

1. Add or remove menus from the menu bar.
2. Customize the content of individual menus.

✔ **Keyboard Shortcuts Customization:** Quickly execute commands using keyboard combinations.

- **Keyboard shortcuts:** Combinations of keys that can be pressed to quickly perform specific actions. You can customize these shortcuts to use your preferred key combinations for frequently used commands.
- **Function of keyboards shortcuts customization**

1. Assign custom shortcuts to frequently used commands.
2. Modify existing shortcuts.
3.Create shortcut sets for different tasks or workflows.

- **The predefined keyboard shortcuts options:**

Ctrl+1 . . . . Front Orientation
Ctrl+2 . . . . Back Orientation
Ctrl+3 . . . . Left Orientation
Ctrl+4 . . . . Right Orientation
Ctrl+5 . . . . Top Orientation
Ctrl+6 . . . . Bottom Orientation
Ctrl+7 . . . . Isometric Orientation
Ctrl+8 . . . . View Normal To
Spacebar . . . . View Orientation dialog
Ctrl . . . . Hold and select multiple items with mouse button
Ctrl+Tab . . . . .Switch between documents

✔ **Mouse Gestures Customization:** Perform actions using specific mouse movements.

- **Mouse gestures:** Combinations of mouse movements and clicks that can be used to perform actions. You can customize these gestures to create shortcuts for specific tasks or to navigate the application more efficiently.
- **Functions of Mouse gestures:**

  1.Define gestures for zooming, panning, and rotating views.

  2.Create custom gestures for specific commands or tasks.

  3.Mouse buttons operate in the following ways:

  4.Left: Selects menu items, entities in the graphics area, and objects in the feeature Manager design tree.

  5.Right: Displays the context-sensitive shortcut menus.

  6.Middle: Rotates pans, zooms a part or an assembly, and pans in a drawing.

✔ **Command Manager Customization:** Manage and organize commands within SolidWorks.

- **Command manager:** A central repository of commands and their associated actions. You can customize the command manager by creating new commands, modifying existing ones, and assigning them to different menus, toolbars, or keyboard shortcuts.
- **Functions of command manager**

  1.Create new commands or modify existing ones.

  2.Assign commands to different toolbars or menus.

  3.Organize commands into categories or groups.

✔ **Shortcut Bars Customization:** Provide quick access to frequently used commands

- **Shortcut bars:** Customizable bars that can be placed on the application's interface to provide quick access to frequently used commands. You can customize the content and arrangement of these bars.

  Examples:

  1. Create custom shortcut bars with specific commands.
  2. Customize the appearance and arrangement of shortcut bars.
  3. Add or remove shortcut bars from the interface.

✔ **Custom Colours and Themes Customization**: Change the visual appearance of SolidWorks.

- **Custom colors and themes:** The appearance of the application's interface can be customized using different color schemes and themes. You can choose from predefined themes or create your own to match your preferences.

  Examples:

  1.Choose from predefined color schemes or create custom ones.

  2.Apply different themes to change the overall look and feel.

  3.Customize individual elements like backgrounds, fonts, and icons.

- ✔ **Workspace Customization:** Arrange windows, panels, and other elements to suit your workflow.
- **Workspace:** The layout and arrangement of windows, panels, and other elements within the application. You can customize your workspace to optimize your workflow and improve productivity.
  Examples:
1. Dock or undock windows.
2. Resize and reposition windows.
3. Create custom workspaces for specific tasks or projects.

- ✔ **File Locations Customization:** Specify default locations for saving and opening files.
- **File locations:** The default locations for saving and opening files. You can customize these locations to specify your preferred directories for different types of files.
  Examples:
  1. Set custom default folders for different file types (e.g., parts, assemblies, drawings).
  2. Create project folders to organize related files.
- ✔ **Custom Property Tab Builder Customization:** Create and customize property tabs for custom features or components.
- ✔ **Custom property tab builder:** A tool that allows you to create and customize property tabs within the application. You can use this tool to define the properties that are displayed and how they are arranged.
- **Functions of Custom property tab builder**
1. Define custom properties and their data types.
2. Set default values for properties.
3. Create custom property tabs with specific layouts.

Practical Activity 1.3.2: Customizing SolidWorks customer Interfaces

Task:

1: As computer technician you are tasked to customize a SolidWorks interfaces Such as toolbars and Menus, Keyboard Shortcuts, Mouse Gestures ,Command Manager Customization ,Shortcut Bars ,Custom Colors and Themes Workspace Customization ,File Locations and Custom Property Tab Builder

2: Read the steps from key reading 1.3.2 in trainee ‘manual

3:Follow the demonstrated steps from trainer of how to customize each SolidWorks Interfaces.

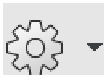
4: Customize SolidWorks Software interfaces by simulating the shown demonstration.

5: Ask for support if there is an occurred issue

Key readings 1.3.2: Customizing SolidWorks Interfaces

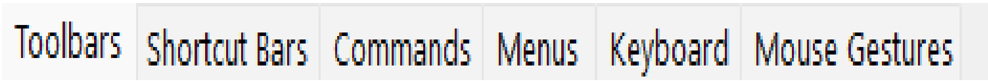
- **Customizing Toolbars and Menus**

  ✔ **Steps:**

  1. Click on down ward arrow on the right side of Setting 

  2. Click on customize

  3. Dialog box will appear with the following titles:



4.Then hover your cursor to each one you want to customize

- **SolidWorks Command Manager with customizable tabs**

  ✔ **Steps:**

  1) Right-click on a tab in the Command Manager.
  2) Choose "Customize..."
  3) Select the "Command Manager" tab.
  4) Add, remove, or rearrange commands within the tabs.
  5) Create new tabs: Click the "New" button to create a new tab.

- **SolidWorks shortcut bar with customizable icons Steps:**

  1) Right-click on an empty area of the shortcut bar.
  2) Choose "Customize..."
  3) Select the "Shortcut Bars" tab.
  4) Add, remove, or rearrange commands on the shortcut bar.

- **Custom Colors and Themes customizing**

  ✔ **Steps:**

  1) Go to Tools > Options > System Options.
  2) Select the "Colors" tab.
  3) Choose a predefined color scheme or create your own.

- **Workspace Customizing**

  ✔ **Steps:**

  1) Go to View > Workspaces.
  2) Choose a predefined workspace or create your own.
  3) Customize the workspace by adding or removing toolbars and panels.
  4) SolidWorks Options dialog box with the File Locations tab

- **File Locations**
  - ✔ **Steps**
    1) Go to Tools > Options > System Options.
    2) Select the "File Locations" tab.
    3) Set the default location for saving files.
- **Custom Property Tab Builder**
  - ✔ **Steps:**
    1) Go to Tools > Options > System Options.
    2) Select the "Custom Property Tab Builder" tab.
    3) Create a new custom property tab.
    4) Add custom properties to the tab.

Points to Remember

✔ **The SolidWorks user Interface is primarily comprised of nine major sections:**

1. The Menu Bar
2. The SOLIDWORKS Menus
3. The Quick Access Tools
4. The Command Manager
5. The Feature Manager Design Tree
6. The Heads-Up View Toolbar
7. The Graphics Area
8. The Task Pane
9. The Status Bar

- Trainee must know the steps of customizing the following elements:

Toolbars and Menus, Keyboard Shortcuts, Mouse Gestures ,Command Manager Customization ,Shortcut Bars ,Custom Colors and Themes Workspace Customization ,File Locations and Custom Property Tab Builder

Application of learning 1.3.

ABD computer peripheral manufacturing company ltd, wants to hire a technician who is a professional worker of CAD Software such as SolidWorks. As a technician you are hired to customize SolidWorks software before starting to draw by using Solid works. An task includes ensuring toolbars and Menus, Keyboard Shortcuts, Mouse Gestures, Command Manager interfaces are well customized.

Learning outcome 1 end assessment

Written assessment

Part I. Read carefully the following statements and choose the correct answer.

1. Which of the following SolidWorks document is used to design individual 3D components?
   a) Template (.slddrw)
   b) Sketch Document (.sldsk)
   c) Part Template (.sldprt)
   d) Assembly Template (.sldasm)
2. **What is the primary function of toolbars in SolidWorks?**
   a) To manage file locations
   b) To display detailed properties of parts
   c) To create assembly drawings
   d) To contain icons for frequently used commands
3. **Which of the following can be the purpose of customizing the menus of SolidWorks?**
   a) The size of the icons
   b) The number of commands in the Command Manager
   c) The color of the application interface
   d) The arrangement and content of menu items
4. **What do keyboard shortcuts allow you to do in SolidWorks?**
   a) Quickly execute commands using key combinations
   b) Manage file locations
   c) Customize the appearance of toolbars
   d) Change the layout of the workspace
5. **Which of the following keyboard shortcuts is used for front orientation?**
   a) Ctrl+7
   b) Ctrl+5
   c) Ctrl+2
   d) Ctrl+1
6. **What is the purpose of mouse gestures in SolidWorks?**
   a) To perform actions using specific mouse movements
   b) To organize commands in the Command Manager
   c) To display context-sensitive menus
   d) To create custom toolbars
7. **How can you customize the Command Manager in SolidWorks?**
   a) By rearranging the menu bar
   b) By creating new commands and modifying existing ones

c) By changing its color scheme
d) By adjusting the workspace layout

8. **Which customization option allows you to change the visual appearance of SolidWorks?**
   a) Shortcut Bars
   b) File Locations
   c) Custom Colors and Themes
   d) Command Manager
9. **What can you customize regarding file locations in SolidWorks?**

   a) The number of toolbars
   b) The layout of the Task Pane
   c) The color of the Command Manager
   d) Default locations for saving and opening files

10. **What is the purpose of the Custom Property Tab Builder in SolidWorks?**
    a) To manage file locations
    b) To customize keyboard shortcuts
    c) To create and customize property tabs for features or components
    d) To arrange toolbars
11. **Which of the following is NOT a major section of the SolidWorks User Interface?**
    a) Status Bar
    b) Menu Bar
    c) Graphics Area
    d) File Manager

Part 2.

1. Match column term and definition in column of answer by using corresponding letter.

Answer	**TERM**	**DEFINITION**
1 ……………..	1. CAD (Computer-Aided Design)	a) Creating designs that show length, width, and depth, giving objects a realistic appearance.
2 ……………..	2. SolidWorks	b) Creating flat drawings with length and width, without depth.
3……………..	3. 2DModeling	c) A software used for 3Dmodeling, simulation, and product design.
4……………..	4. 3DModeling	d) A system used for creating, modifying, and optimizing designs digitally.

5……………	5.CAE	e) Features, Sketch, Evaluate, View, SolidWorks Resources

2. Match Column A and B in column of Answer by using corresponding letter.

ANSWERS	Column A	Column B
1**)...**	1) SolidWorks templates include ,………..	a) User-friendly, cost-effective, parametric design, wide industry use
2)...	2) Tabs that appear when you first launch SolidWorks,…….	b) Command Manager, Feature Manager Tree, Graphics Area, Menu Bar
3)...	3) Examples of SolidWorks user interfaces…..	c) Part ,Assembly, Drawing
4)…	4) Advantages of using SolidWorks in architecture and computer systems, ………..	d) Features, Sketch, Evaluate, View, SolidWorks Resources

3. Match the correct descriptions in following column A and column B in column C

Column A	Column B	Column C
1) .sldprt	a) A widely recognized drawing standard used internationally	1) …
2) .sldasm	b) A drawing standard used predominantly in the U.S.	2) …
3) .slddrw	c) A native file format for parts in SolidWorks	3) …
4) ISO	d) The largest ISO paper size (841 x 1189 mm)	4) …
5) ANSI	e) A file format used for assemblies of multiple parts	5) …

Part 3.Read the following statement and choose the correct answer (multiple selections are allowed)

Q1. Customizing keyboard shortcuts in SolidWorks helps to:

a) Enhance visual design

b) Improve access to frequently used commands

c) Increase mouse usage

Q2. Match the benefits of using keyboard shortcuts in SolidWorks:

a) Slows down workflow

b) Speeds up design tasks

c) Improves precision

Q3. Match the benefits of customizing the Command Manager in SolidWorks:

a) Reduces screen clutter

b) Makes tools harder to find

c) Organizes tools for easier access

d) Personalizes the workspace layout

Practical assessment

ABD ltd wants to install SolidWorks software into their computers for better producing CAD based Computer peripherals at their company, as CAD professional ,you are asked to install SolidWorks software .Task includes the installation of SolidWorks Software version 2023, creating template and customizing the interfaces.

References

1.Mackay, R. (den 6 February 2021). How to Install SOLIDWORKS Desktop Software. *SolidWorks TECH TIPS*. https://www.javelin-tech.com/blog/2021/02/how-to-install-solidworks-software/

2.Ji, P. (2011). *SolidWorks essentials*. Concord, Massachusetts: Ji Pengcheng https://www.scribd.com/doc/194529688/SolidWorks-Essentials-Ver-2011

Learning Outcome 2: Create Part

Indicative contents
2.1. Selecting of plane **2.2. Creating sketch** **2.3. Applying part features** **2.4. Applying Visualization** **2.5. Exporting part**

Key Competencies for Learning Outcome 2: Create Part

Knowledge	Skills	Attitudes
✔ Selection of Plane in SolidWorks software ✔ Application of part features in SolidWorks software ✔ Application of visualization in SolidWorks software	✔ Selecting Plane in SolidWorks software ✔ Creating Sketch in SolidWorks software ✔ Applying Part Features in SolidWorks software ✔ Applying Visualization in SolidWorks software ✔ Exporting part in SolidWorks software	✔ Taking initiative ✔ Having Creativity ✔ Being Critical thinker ✔ Being Problem Solver ✔ Having Passion ✔ Having Adaptability ✔ Being Decision Maker ✔ Being Team worker

Duration: 20 hrs

Learning outcome 2 objectives:

By the end of the learning outcome, the trainees will be able to:

1.Select properly SolidWorks software Standard Plane and custom plane based on assigned. task.

2. Create appropriately a Sketch in SolidWorks software based on related task.

3.Define different Features tools use in SolidWorks software.

4.Apply correctly Visualization in SolidWorks software related to the work.

5.Export properly part in SolidWorks software refer to the acquired file extension format.

Resources

Equipment	Tools	Materials
✔ Computer ✔ Projector ✔ Whiteboard	✔ SolidWorks software ✔ Storage media devices ✔ SolidWorks tutorials	✔ Marker pen ✔ A4 Papers or flip chart ✔ Internet connectivity

Indicative content 2.1: Selecting of Plane

Duration: 3hrs

Theoretical Activity 2.1.1: Selection of SolidWorks Planes

Tasks:

1:Answer the following questions refer to the introduction:

i. What are the types of planes made up SolidWorks?
ii. What are the commonly used standard planes in SolidWorks?
iii. what are the types of references necessary to define custom plane?
iv. Identify the steps to be followed for creating custom planes to suit your specific design needs?
v. What is the importance of defining the correct orientation of a plane in SolidWorks ?

2: Write the findings on paper or flip chart

3: Present the findings/answers on paper/flip chart

4: Pay attention on trainer's clarifications and ask questions if any.

5: For more clarification, read the key readings 2.1.1 in Trainee manual

Key readings 2.1.1: Selection of SolidWorks Planes

Selecting the appropriate plane is crucial for starting your design.

- SolidWorks Software is made up with two types of planes:
  - ✔ Standard plane
  - ✔ Custom plane
- Standard plane is contained by 3 default planes such as:
  - Front plane: The front plane is the reference plane that represents the front-facing view of your part. It is typically used as the starting point for sketching features that are visible from the front of the part.
  - Top plane: The top plane is the reference plane that represents the top or plan view of your part. It is often used as the starting point for sketching features that are visible from the top of the part.
  - Right plane: The right plane is the reference plane that represents the right-side view of your part. It is commonly used as the starting point for sketching features that are visible from the right side of the part.
- **Here are the commonly used standard planes:**
  - ✔ **Standard plane**

SolidWorks provides standard planes that help define the orientation and position of your part In addition to the standard planes provided in SolidWorks, you also have the option to create custom planes to suit your specific design needs.

✔ **Custom plane**

Custom planes allow you to define planes that are oriented at different angles or positions, providing flexibility in your part creation process.

- **Selection of references for Plane Creation:**
  - ✔ To define a custom plane, SolidWorks needs at least one reference. You can use the following
  - ✔ **Types of references:**
    - A face
    - A plane (existing standard planes like Front, Top, Right)
    - An edge
    - A point
    - A sketch

Practical Activity 2.1.2: Selecting planes

Task:

1. Refer to the introduction you are asked to perform the following task:
   As computer technician you are asked to select standard planes and after that create your own custom plane in SolidWorks.
2. Read the steps from key reading 2.1.2 in trainee' manual.
3. Pay attention to your trainer on how to select plane in SolidWorks software.
4. Select plane in SolidWorks Software by simulating your trainer.
5. Ask for clarification from trainer if any.

Key readings 2.1.2. Selecting planes

- **Selecting standard planes Steps:**

  1. Start by opening a new part file in SolidWorks.

  2. Access the Feature Manager Design Tree

  3. Locate the Standard Planes in the Feature Manager

  4. Select the Desired Plane by clicking on one of the planes (Front, Top, or Right) depending on the orientation you want for your sketch.

5. Start a New Sketch by click on the Sketch tab on the Command Manager toolbar

6. View Normal to Plane and choose Normal To from the context menu.

7. Begin Sketching use sketch tools to create 2D geometry on the plane.

- **Creating a custom plane steps:**

1. Open a part or assembly document in SolidWorks.

2. Go to the "Features" tab in the Command Manager.

3. Click on the "Reference Geometry" dropdown menu and select "Plane."

4. A dialog box will appear, allowing you to define the orientation and position of the custom plane.

5. Choose the reference entities or coordinate system that will determine the location and orientation of the plane.

6. Specify the distance or angle from the reference entities to position the plane accurately.

7. Click "OK" to create the custom plane.

Points to Remember

- **SolidWorks Software is made up with two types of planes:**

1. Standard plane
2. Custom plane

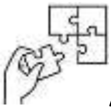
✔ **Standard plane** is contained by three (3) default planes such as**:**

- Front plane
- Top plane
- Right plane

- **Custom plan**: Is created when they want to draw multi complex drawings.
  - ✔ While selecting standard planes steps trainee must know the steps to be followed
  - ✔ While creating a custom plane steps trainee must know the steps to be followed
  - ✔ Trainee must know the Types of references during the creation of custom plane

    - A face

- A plane (existing standard planes like Front, Top, Right)
- An edge
- A point
- A sketch



Application of learning 2.1.

ABD company ltd wants to start drawing a new products by using SolidWorks software, As professional SolidWorks user you are asked to select default planes and create a custom plane for complying to the design.

Indicative content 2.2: Creating Sketch

Duration: 6hrs

Theoretical Activity 2.2.1: Creation of Sketch

Tasks:

1: Read and answer the following questions:

i. What do you understand by the following terms 2D sketch in SolidWorks?
ii. Identify the key Elements of a Sketch.
iii. Describe sketch tools and sketch entities in SolidWorks?
iv. Discuss on sketch relations in SolidWorks?
v. Describe sketch status in SolidWorks?

2: Write your findings on papers/flipcharts.

3: Present your findings.

4: Pay attention on trainer's clarifications and ask questions if any.

5: For more clarification, read the key readings 2.2.1 in trainee manual

Key readings 2.2.1.: Description of sketch

- **2D sketch**

When creating parts in SolidWorks, 2D sketches are used as the foundation for defining the shapes and features of the part 's geometry. A 2D sketch is the starting point for creating a part in SolidWorks. It is a 2-dimensional representation of the part's geometry and features. You can create a 2D sketch on a selected plane or face of the part.

There are various ways of creating a sketch. All sketches include the following elements: Origin plane and sketch entities

✔ **Origin:** In many instances, you start the sketch at the origin, which provides an anchor for the sketch.

✔ Sketch entities

SolidWorks provides a variety of sketch tools to help you create and modify your sketches. These tools include lines, arcs, circles, rectangles, splines, Move Entities, Rotate Entities , Offset Entities and more. You can use these tools to sketch the desired shape and dimensions of your part. Specifically sketch entities are used to edit or modify design intent.

Icons	Name
	Construction Geometry
	Text
	Plane
	Sketch Fillet
	Sketch Chamfer
	Offset Entities
	Offset On Surface
	Convert Entities
	Intersection Curve
	Face Curves
	Segment
	Trim Entities
	Extend Entities
	Split Entities
	Mirror Entities
	Dynamic Mirror Entities
	Move Entities
	Rotate Entities
	Scale Entities
	Copy Entities
	Replace Entity
	Stretch Entities
	Linear Sketch Pattern
	Circular Sketch Pattern
	Make Path
	Modify Sketch

	No Solve Move
	Sketch Picture
	Sketch Numeric Input
	Sketch Dimension Driven
	Add Dimension

- **Sketch Tools**

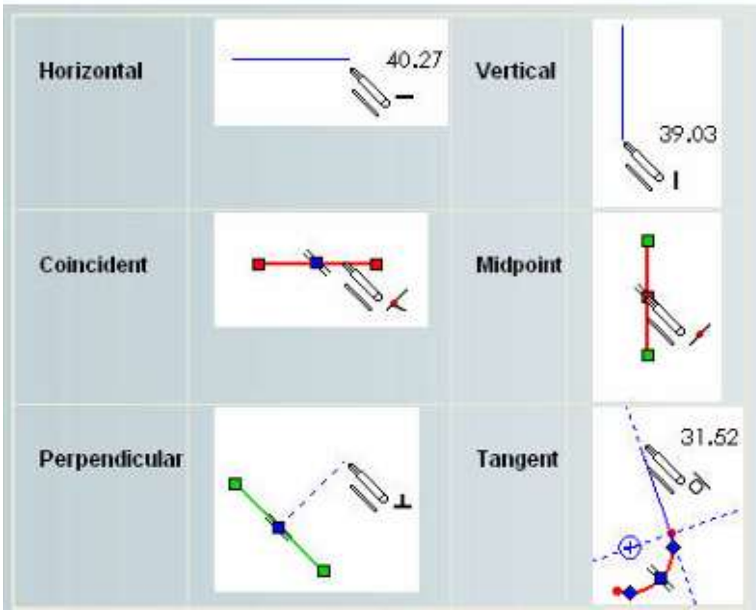
  Sketch tools are the basic sketch creation tools that used to create a new sketch. They include lines, arcs, circles, ellipses, polygon and Smart dimension. You can combine and manipulate these entities to create complex shapes and features.

Icons	**Name**
	Select
	Grid/Snap
	Sketch or Exit Sketch
	3D Sketch
	3D Sketch On Plane
	Slicing
	Rapid Sketch
	Instant2D
	Shaded Sketch Contours
	Line
	Corner Rectangle
	Center Rectangle
	3 Point Corner Rectangle
	3 Point Center Rectangle
	Parallelogram
	Straight Slot
	Centerpoint Straight Slot
	3 Point Arc Slot
	Centerpoint Arc Slot

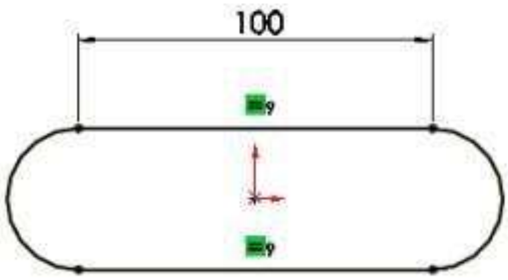
	Polygon
	Circle
	Perimeter Circle
	Center point Arc
	Tangent Arc
	3 Point Arc
	Ellipse
	Partial Ellipse
	Parabola
	Conic
	Spline
	Style Spline

- **Sketch relations**

  Sketch relations are used to define the geometric relationships between sketch entities. They ensure that the sketch maintains its intended shape and dimensions. Examples of sketch relations include horizontal/vertical alignment, tangency, symmetry, concentricity, and parallelism.



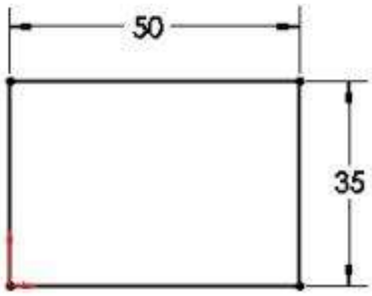
The green symbols indicate that there is an equal relation between the horizontal lines:



✔ Status of a Sketch

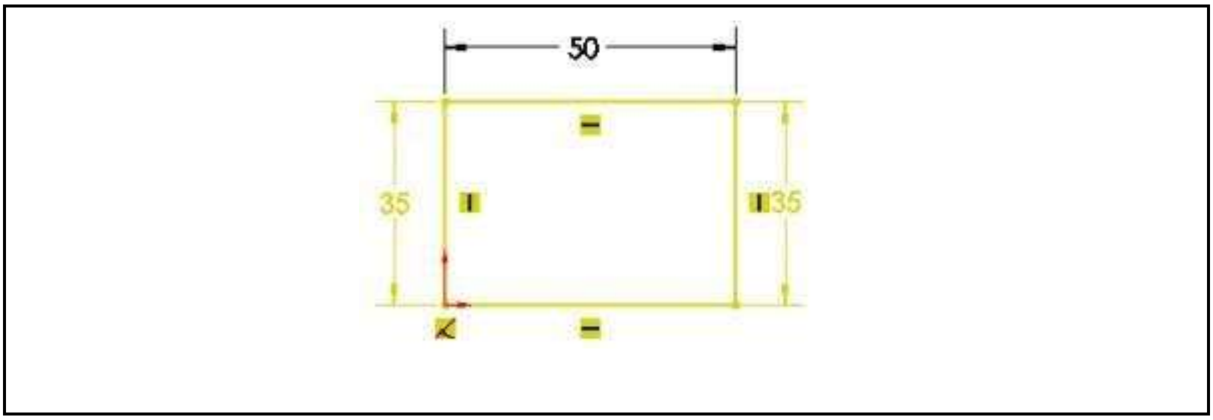
The status of a sketch refers to how well-defined the sketch is in terms of its geometry and dimensions. Understanding the status of a sketch is important for maintaining control over the design and ensuring the accuracy and stability of the part. By properly defining the sketch with dimensions and relations, you can effectively control the shape and behavior of the part. In SolidWorks, the status of a sketch can be classified **as fully defined, under defined**, or **over defined.**

- **Fully defined:** A fully defined sketch has all its dimensions and geometric relationships specified, leaving no degrees of freedom. This means that the sketch is completely constrained and its shape is fixed.



- **Under defined:** An under defined sketch has some dimensions or geometric relationships that are not fully specified. This results in some degrees of freedom, allowing the sketch to change its shape or position. It is important to fully define a sketch to ensure its stability and accuracy.

- **Over defined:** An over defined sketch has redundant or conflicting dimensions or geometric relationships. This can lead to errors or unexpected behavior in the sketch. It is necessary to remove the redundant constraints or dimensions to achieve a fully defined or under defined status.



Practical Activity 2.2.2: Creating Sketch

Task:

1. Refer to the introduction perform the activity of the following task:
   As computer technician you are asked to select a front plane and create sketch from center point Straight Slot of 100 mmx80 mm and after that make sure the sketch geometry is full defined.
2. Read the steps from key reading 2.2.2 in Trainee's manual.
3. Pay attention on demonstration from trainer on how to create sketch in SolidWorks.
4. Create sketch by simulating what you have seen from trainer's demonstration.
5. Ask for clarification if there is any occurred issue

Key readings 2.2.1 : Creating Sketch

- **Steps:**

1. Launch SolidWorks and create a new part file.
2. Choose a plane (e.g., Front, Top, or Right)
3. Click on the "Sketch" tab in the Command Manager, then select "Sketch" to enter sketch mode.
4. Select the desired sketch tools (e.g., Line, Circle, and Rectangle) from the Sketch toolbar to create your geometry.
5. Click on the workspace to place points, lines, or shapes.
6. Apply Dimensions and Relations
7. Modify sketch entities as needed by selecting and dragging them or using the properties toolbar.
8. Click "Exit Sketch" to return to the part environment.
9. Save the part file to ensure your sketch is retained.

Application of learning 2.1

✔ **2D Sketch**:

- Starting point for part creation.
- Represents the part's geometry in two dimensions.
- Can be created on a selected plane or face.

✔ **Key Elements of a Sketch**:

- Origin
- Plane
- Sketch Entities

✔ **Sketch entities available for creating and modifying sketches:**

- Move Entities
- Rotate Entities
- Copy Entities

- Split Entities
- Scale Entities
- Offset Entities
- Stretch Entities
- Convert Entities
- Trim Entities
- Stretch Fillet
- Mirror Entities
- Make Path

✔ **Sketch Tools** are tools that used for creating Basic geometric shapes include:

- Line, Corner Rectangle, Center Rectangle, Parallelogram
- Circle, Arc, Polygon, Spline, Text, Centerline, Midpoint Line…

✔ **Sketch Relations Examples**: Horizontal/Vertical Alignment, Tangency, Symmetry, Concentricity, Parallelism.

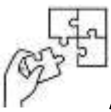
✔ **Status of a Sketch**:

- **Fully Defined**
- **Under Defined**
- **Over Defined**

✔ **Rule of Thumb When Sketching**:

- Create a rough sketch.
- Add relations to define relationships.
- Add dimensions for accuracy.

✔ You must know the steps of creating sketch in solidWorks:



Application of learning 2.4.

XZ Company needs a technician for creating sketches in SolidWorks at the beginning of their 3D design model of their new product. As a technician of Computer system and Architecture create a 2d sketch by using sketch a Corner Rectangle, Center Rectangle, 3 Point Corner Rectangle, and a 3 Point Center Rectangle width each one has width of 20mm and length of 40mm. ensure that sketch status is fully defined and sketch relation are applied.

Indicative content 2.3: Applying Part Features

Duration: 5 hrs

Theoretical Activity 2.3.1: Description of each SolidWorks feature tool

Tasks:

1. In small groups discuss on the following questions:

   i. Discuss on the classification of sketched feature and applied feaures in Solid Works?

   ii. Describe the different SolidWorks part feature tools ?

   iii. Highlight the examples of SolidWorks feature tool end conditions.

2. Write the findings on paper or flip chart
3. Present the findings/answers on paper/flip chart
4. Pay attention on trainer's clarifications and ask questions if there is any.
5. For more clarification, read the key readings 2.3.1 in trainee manual

Key readings 2.3.1: Description of SolidWorks feature tools

- **Features** can be classified as either sketched or applied.
  - **Sketched Features:** One that is based upon a 2D sketch only this type can be used as a base feature.
  - **Applied Features**: Created directly on the solid model. Fillets and chamfers are examples of this type of feature.

✔ **SolidWorks Feature types are:**

- Basic features,
- Modifying features and
- Pattern features

✔ **Basic Features types**

1) **Extruded Boss:** Projects a two dimensional sketch linearly to create a three dimensional part. Can be two directional and Creates a solid feature by extending a 2D sketch profile along a straight path.

2) **Revolved Boss:** Creates a solid feature by rotating a 2D sketch profile around an axis.

3) **Swept Boss:** Creates a solid feature by sweeping a 2D sketch profile along a curved path.

4) **Lofted Boss:** Creates a solid feature by connecting two or more 2D sketch profiles along a curved path.

5) **Extruded Cut:** Removes material from a solid feature by extending a 2D sketch profile along a straight path.

6) **Revolved Cut:** Similar to the revolve boss/base feature. Revolve cut takes a two dimensional sketch and revolves it about an axis to remove material from a preexisting part.

- ✔ **Modifying Features types**

1) **Fillet:** Rounds off sharp corners, or adds volume to the material by rounding off an inner corner. The user inputs the radius of the fillet. This feature does not require a sketch to use; only a preexisting part is needed.

2) **Chamfer** can be found under the pull down arrow. This feature performs similarly except that it breaks the edges rather than rounds them.It Creates a beveled edge between two or more faces.
3) **Rib:** An extrude feature created from open or closed sketch contours. The user specifies the desired thickness and the directional path between the sketch contour and the part/Creates a thickened feature that extends from one face to another.

4) **Shell:** Creates a hollow feature by removing material from the outer surface of a solid.

- **Pattern Features types**

1) **Linear Pattern:** Creates multiple instances of a feature arranged in a straight line.

2) **Circular Pattern:** Creates multiple instances of a feature arranged in a circle.
3) **Mirror :** Creates a mirror image of a feature across a plane.

- **Examples of design intent in respect to End Conditions of features** :
  - Blind
  - Through all
  - Mid-plane
  - Up to Next
  - Up to Surface

Practical Activity 2.3.2: Applying part features

Task:

1. Refer to the introduction of the activity you are asked to perform the following task:

As computer technician you are asked to create 25 CD storage box ,By following design criteria for a CD storage box ,Overall size for 10 CD storage box = 25cm x 12.4cm x 14.2cm The CD storage box is constructed of a polymer (plastic) material.

2. Read the steps from key readings 2.3.2 in trainer manual

3. Follow the demonstration from your trainer on how to apply some SolidWorks features

4. Apply feature by simulating trainer what have shown in demonstration.

5. Ask for support from your trainer if there is an occurred issue.

Key readings 2.3.2: Applying part features

- **Applying part features**

✔ **Extruded Boss/Base Steps**

1.Create a 2D sketch on a plane.
2.Select the sketch and the "Extruded Boss/Base" command.
3.Define the extrusion direction and depth.
4.Set the end condition (blind, through all, up to next).
5.Adjust other settings (draft, fillet, chamfer).
6.Click "OK" to create the feature.

✔ **Revolved Boss Steps:**

1. Create a 2D sketch on a plane.
2. Select the sketch, the "Revolved Boss" command, and the axis of rotation.
3. Define the angle of rotation.
4. Adjust other settings (draft, fillet, chamfer).

✔ **Swept Boss Steps:**

1. Create two 2D sketches on different planes.
2. Select the sketches, the "Swept Boss" command, and the path.
3. Adjust other settings (fillet, chamfer).
4. Click "OK" to create the feature

✔ **Lofted Boss Steps:**

1. Create multiple 2D sketches on different planes.
2. Select the sketches, the "Lofted Boss" command, and the path.
3. Adjust other settings (twist, draft, fillet, chamfer).
4. Click "OK" to create the feature.

- **Applying modifying feature**

✔ **Fillet Steps:**

1. Select the **Fillet** tool from the Features tab.
2. Click on the edges or faces where you want to apply the fillet.
3. Specify the radius of the fillet.
4. Confirm the selection to create the fillet.

✔ **6.Chamfer Steps:**

1. Select the **Chamfer** tool from the Features tab.
2. Choose the edges to be chamfered.
3. Set the chamfer type (distance and angle or two distances).
4. Click to apply the chamfer.

✔ **Rib Steps:**

1. Select the **Rib** tool from the Features tab.
2. Create or select a 2D sketch for the rib profile.
3. Specify the thickness and direction of the rib.
4. Click to create the rib feature.

✔ **Shell Steps:**

1. Select the **Shell** tool from the Features tab.
2. Choose the solid body you want to hollow out.
3. Specify the wall thickness.
4. Select faces to remove if necessary.
5. Confirm to apply the shell feature.

- **Applying modifying feature**

✔ **Linear Pattern Steps:**

1. Select the **Linear Pattern** tool from the Features tab.
2. Choose the feature or body to pattern.
3. Define the direction and spacing for the pattern.
4. Specify the number of instances in each direction.
5. Click to create the linear pattern.

✔ **Circular Pattern Steps:**

1. Select the **Circular Pattern** tool from the Features tab.
2. Choose the feature or body to pattern.
3. Select the central axis for the circular arrangement.
4. Define the number of instances and angle between them.
5. Click to apply the circular pattern.

✔ **Mirror Steps:**

1. Select the **Mirror** tool from the Features tab.
2. Choose the features, faces, or bodies you want to mirror.
3. Select the mirror plane (face or plane).
4. Confirm to create the mirrored geometry.

Points to Remember

- ✔ Trainees must know to create 2D sketch and apply feature
- ✔ Trainees must know the following features Basic Features types
    - Extruded Boss
    - Revolved Boss
    - Swept Boss
    - Lofted Boss
    - Extruded Cut
    - Revolved Cut
- ✔ Trainer muster know to modifying the exiciting featuire Features.

Application of learning 2.3

AS computer peripherals technician, you've been hired to create a new model of an iPhone enclosure measuring 160.1mm x 77.9mm x 8.37mm and 12 mm of thichness .Finally, use different posible SolidWorks feature tools that can be applied to design this enclosure.

Indicative content 2.4: Applying Visualization

Duration: 3 hrs

Theoretical Activity 2.4.1: Description of visualization tools

Tasks:

1. Refer to the introduction of the activity in small groups you are asked to answer the following questions:
   i. Define visualization?
   ii. Describe different part visualization tools that are found in Solid Works
   iii. Specify the purpose of each part visualization tools?
   iv. Why is view orientation crucial when creating models in SolidWorks, and how can it affect the ease of navigation and clarity of presentations?
   v. How do different display styles, such as shaded ,wireframe, contribute to understanding a model's geometry during the design process?

2. Write your findings on papers/flipcharts.
3. Present your findings on papers/flipcharts.
4. Follow the expert view from trainer and ask clarification if there is any.
5. Read the Key readings 2.4.1 in trainee manual.

Key readings 2.4.1. Description of Visualization tools

✔ **Definition of Visualization**

Visualization refers to the process of creating **realistic representations** of 3D models or parts using various tools such as materials, appearances, lighting, and camera settings. The goal of visualization is to help users and stakeholders understand what a final product will look like, how it will behave under certain conditions, and how it will interact with its environment.

- **Visualization tools of SolidWorks :**

1.Material

✔ **Definition**:

- Materials in SolidWorks represent the physical characteristics of the part, affecting both its appearance and physical properties such as strength, density, and thermal conductivity.

✔ **Purpose**:

- Visual Representation: The material determines the look of the part, including its color, texture, and reflectivity.
- Physical Simulation: Materials are vital for accurate simulation in stress analysis, thermal performance, and other engineering evaluations.

2. Appearance

✔ **Definition**:

- Appearance in SolidWorks defines the visual look of a part, including color, texture, and transparency, but does not affect physical properties like materials do.

✔ **Purpose**:

- Enhancing Visuals: Appearance allows you to adjust how a part looks without changing its physical attributes, useful for presentation and rendering.
- Customizing Designs: It enables customization of the part's surface finish, whether polished, rough, matte, or glossy.

3. Scene and Environment

✔ **Definition**:

- A scene in SolidWorks includes backgrounds and lighting settings that affect how parts and assemblies are displayed in the viewport or during rendering. The environment sets the context in which your model is visualized.

✔ **Purpose**:

- Realistic Rendering: By applying different scenes, you can make your parts appear as if they are in real-world settings (e.g., studio lighting, outdoor scenes).

- Light Behavior: Environments influence how light interacts with your part, adding realism to the visualization.

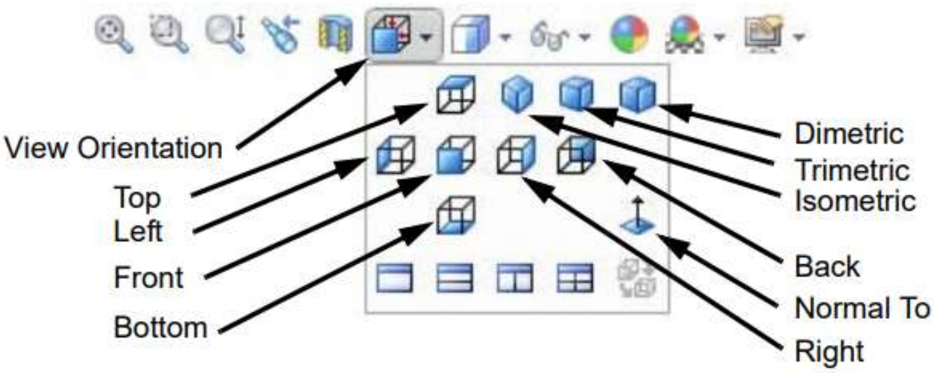
4. View Orientation

✔ **Definition**:

- View orientation controls how the part is positioned and viewed in the 3D workspace.

✔ **Purpose**:

- Ease of Navigation: It allows you to switch between standard views like top, front, isometric, or custom views to examine the model from various angles.
- Presentation: Proper view orientation is essential for creating clear and communicative drawings or presentations of the model.



5. Display Style

✔ **Definition**:

- Display style refers to how the model is visually represented in terms of edges, lines, and shading. It includes modes like wireframe, shaded, and hidden lines visible.

✔ **Purpose**:

- Clear Visualization: Different display styles help in analyzing the model's geometry, edge quality, and internal structure.
- Technical Clarity: Useful for technical drawings where different representations are needed (e.g., hidden lines for assembly views).

✔ **Types of Display Styles**:

- **Shaded:** Fully shaded model with or without edges.

- **Wireframe:** Only the edges of the model are visible.
- **Hidden Lines Removed/Visible:** Displays the model with hidden lines either removed or shown.

6. Lighting

✔ **Definition**:

- Lighting in SolidWorks simulates how light sources interact with the part, affecting how it is rendered in terms of brightness, shadows, and reflections.

✔ **Purpose**:

- Realistic Rendering: Proper lighting enhances the realism of rendered parts, highlighting details, surface finishes, and creating shadows.
- Depth and Dimension: Lighting gives depth to parts, making them appear more three-dimensional and life-like.

✔ **Types of Lighting**:

- Ambient: General lighting that illuminates all parts evenly.
- Directional: Simulates sunlight or other directed light sources.
- Spotlight and Point Light: Focused lighting for detailed highlights.

7. Camera Setting

✔ **Definition**:

- Camera settings allow you to create virtual cameras within the SolidWorks environment to define specific view angles, focal lengths, and perspective for visualizing the part.

✔ **Purpose**:

- Control Over Perspective: Cameras offer greater control over the perspective and depth of the scene, crucial for presentations, animations, and rendered images.
- Customization: Different camera angles can simulate real-world photography setups for more professional renderings.

8. Evaluate

✔ **Definition**:

- The Evaluate tab in SolidWorks contains tools for analyzing the **visual** and **physical** properties of parts and assemblies. This includes evaluating mass, dimensions, and features.

✔ **Purpose**:

- Quality Check: Use the Evaluate tools to check model dimensions, mass properties, and other metrics to ensure design accuracy.
- Rendering Previews: Evaluate visual properties before running a full render to check how the material, appearance, and lighting will look.

Practical Activity 2.4.2: Applying visualization

Task:

1. Refer to the introduction you are asked to perform the following task:

As computer technician you are asked to create CD storage box 17cm x 12.4cm x 14.2cm and use visualization modes such as material, appearance, Scene and Environment, View orientation and Display style .The CD storage box is constructed of a polymer (plastic) material.

2. Read the steps from key reading 2.4.2 in trainee's manual

3. Follow trainer's demonstration on how to apply visualisation by using some visualisations tools.

4.Simulate how to apply visualisation in SolidWorks Software by referring to assigned task.

5. Ask for clarification related to the any occurred issued.

Key readings 2.4.2: Applying visualization

1. Material

✔ **Steps to Apply**:

1) Right-click the part in the Feature Manager.
2) Select Material > Edit Material.
3) Choose the desired material from the library (e.g., steel, aluminum, plastic).
4) Click Apply to assign the material properties.

2. Appearance

✔ **Steps to Apply**:

1) Right-click the part or face in the graphics area.
2) Select appearance from the context menu.
3) Choose a preset appearance (e.g., polished metal, matte plastic, wood) or customize it with colors and textures.

3.How to Use View Orientation:

Click the View Orientation icon or press the space bar > Choose standard views or create a custom view.

4.How to Adjust Lighting:

Go to Scene settings > Edit Lights > Adjust light intensity, position, and type.

5.Scene and Environment :

✔ **Steps to Apply:**

1) Go to the Scene, Lights, and Cameras tab in the task pane.
2) Right-click Scene and choose Edit Scene.
3) Select a preset scene (e.g., studio, outdoor, or indoor environment) or customize it by adding your own background images and lighting.
4) Adjust the environment rotation and floor reflection for better visualization.

6. View Orientation

✔ **Step to Apply:**

1) Click the View Orientation button in the heads-up toolbar or press the spacebar.
2) Choose from preset views (e.g., Isometric, Front, Right) or create a custom orientation by manually rotating the model.
3) Save the custom orientation for later use if needed.

7.Display Style

✔ **Steps to Apply:**

1) Select the Display Style dropdown from the heads-up toolbar.
2) Choose from options like Shaded with Edges, Wireframe, Hidden Lines Removed, or Hidden Lines Visible.

8. Lighting

✔ **Steps to Apply:**

1) Go to the Scene, Lights, and Cameras tab in the task pane.
2) Right-click Lights and choose Add Light or Edit Light.
3) Adjust the type (e.g., ambient, directional, spot), intensity, and position of lights.
4) You can also enable/disable certain lights to create the desired visual effect.

9. Camera Setting

✔ **Steps to Apply:**

1) Go to the Scene, Lights, and Cameras tab.
2) Right-click Cameras and select Add Camera.
3) Position the camera and set its orientation, field of view (zoom), and depth of field.
4) You can switch to Camera View to see how the model looks from the camera's perspective.

10. Evaluate

✔ **Steps to Apply:**

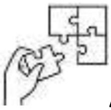
1) Go to the Evaluate tab in the command manager.
2) Tools available include Mass Properties, Section View, Interference Detection, Draft Analysis, and Measure.
3) Use these tools to analyze weight, check for interference in assemblies, or measure critical dimensions.

Points to Remember

- **Visualization tools :**

✔ Material
✔ Appearance
✔ Scene and Environment
✔ View Orientation
✔ Display Style
✔ Lighting
✔ Camera Setting
✔ Evaluate

- Trainee must know the steps of using SolidWorks Software Visualization tools



Application of learning 2.4

As a technician in Computer Systems and Architecture, you are hired to design a CD storage box with dimensions of 54.0 cm in width, 16.4 cm in height, and 17.2 cm in depth. By using SolidWorks, you'll apply visualization techniques on a pre-made part file to enhance your design. This includes applying materials, customizing the appearance, and editing the scene with appropriate backgrounds and lighting. You'll also adjust view orientations to analyse the model from various angles and explore different display styles. Finally, you will use evaluation tools to assess mass properties and dimensions, emphasizing the importance of effective visualization in the design process.

Indicative content 2.5:Exporting Part

Duration:3hrs.

Theoretical Activity 2.5.1: Description of part export in SolidWorks

Tasks:

1. After forming small groups you are asked to answer following questions:
   i. what is export part in SolidWorks
   ii. What are the purpose for exporting parts from SolidWorks?
   iii. What are the most common file formats supported for exporting parts from SolidWorks?
   iv. What are the standard file formats supported for exporting parts from SolidWorks?
2. Write your findings on papers/flipcharts
3. Present your findings/ answers
4. Pay close attention to the expert view from trainer
5. Ask for clarification related to the any occurred issue .

Key readings 2.5.1. Description of part export in SolidWorks

✔ **Definition**

- Exporting a Part in SolidWorks refers to the process of saving a part or assembly model in a different file format than the native SolidWorks format (.sldprt for parts, .sldasm for assemblies) so that it can be used in other applications or shared with others.
- Exporting in SolidWorks involves saving a part or assembly in a different file format for use in other applications or for sharing with others.

✔ **Purpose**

- Facilitate collaboration with team members who may use different software.
- Prepare files for manufacturing, presentation, or documentation.

- Ensure compatibility with various file formats as required by clients or project specifications.

✔ **File format**

- A file format defines how data is structured and stored within a file.
- Choosing the correct format ensures compatibility with SolidWorks and other software.
- SolidWorks provides various file formats for exporting parts, such as prt, sldprt, STEP, IGES, STL, and more. The file format you choose depends on the intended use and compatibility requirements of the exported file.

- **Common SolidWorks Export Formats**:

✔ **.SLDPRT (SolidWorks Part):**Native file format for parts in SolidWorks. It preserves all the parametric design information for further editing in SolidWorks.

- **Use:** For continuing work within SolidWorks.

.**STL (Stereolithography):**A mesh format that converts 3D models into triangles, commonly used for 3D printing.

- **Use:** Ideal for 3D printing, as it simplifies the model into a format printers can read.

✔ **.STEP / .STP (Standard for the Exchange of Product Data):**A neutral, widely-used format for sharing CAD data across different software platforms.

- **Use**: Useful for sharing detailed 3D models with other CAD systems without losing design integrity.

✔ .**IGES / .IGS (Initial Graphics Exchange Specification):**Another neutral file format used for exchanging CAD models, especially surface models.

- **Use:** For transferring both 2D and 3D CAD data between different software platforms.

✔ **.PDF (Portable Document Format):**A 2D format for sharing drawings, allowing others to view or print without needing CAD software.

- **Use**: Best for distributing technical drawings or documents for review or printing.

✔ **.DXF /.DWG**: Standard formats for exchanging 2D drawings and technical designs, often used in AutoCAD.

- **Use:** Common in CNC machining, laser cutting, and when transferring 2D data to other design programs.

✔ **.Parasolid (.X_T**):A format used for exchanging precise geometric models between different 3Dmodeling software.

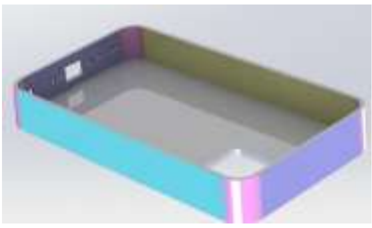
- **Use:** For ensuring accurate data transfer between different CAD systems that support Parasolid files.

- ✔ **File location:** When exporting a part in SolidWorks, you can choose the desired file location where you want to save the exported file. This can be a local folder or a network location.
- ✔ **Methods for Backing Up SolidWorks Files**:
  - **Manual Backups**: Periodically copy files to external drives or different folders.
  - **Automatic Backups**: Utilize software that automates backups at scheduled intervals.
- ✔ **Differences Between Network Drives and Local Storage**:
  - **Network Drives**: Accessible from multiple devices within a network, ideal for collaborative work environments.(Cloud storage options (e.g., Google Drive, Dropbox, OneDrive) provide remote access, easy sharing, and automatic backup solutions.
  - **Local Storage**: Files are stored on a single device, allowing for faster access but limited to that machine.

Practical Activity 2.5.2 Exporting Part

Task:

1 :After forming small groups you are requested to perform the following task
As technician from computer peripheral manufacturing company you are requested to redraw the image below without considering scale and its real size and export it to jpg and pdf



2: Read the steps from **key reading 2.5.2 in trainee** manual
3: Pay attention to trainer's demonstration
4: Export part by following the steps demonstrated by trainer
5: In addition, ask questions where necessary.

Key readings 2.5.2 .Exporting Parts Using SolidWorks Software

Steps to Export a Part :

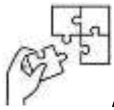
1. Start SolidWorks and open the part file you wish to export.
2. Go to File Menu and Click on File in the menu bar.
3. Choose Save As from the dropdown menu.
4. Choose Export Format In the "Save as type" dropdown menu, select the desired file format (e.g., .STEP, .IGES, .STL).
5. Enter a file name and choose the destination folder.
6. Click on the Options button to configure settings specific to the chosen format (e.g., exporting solid bodies, surfaces, mesh quality for .STL).
7. Click Save to complete the export process.
8. Exporting Options

Points to Remember

- While exporting a Part in SolidWorks trainee must know the steps to be used.

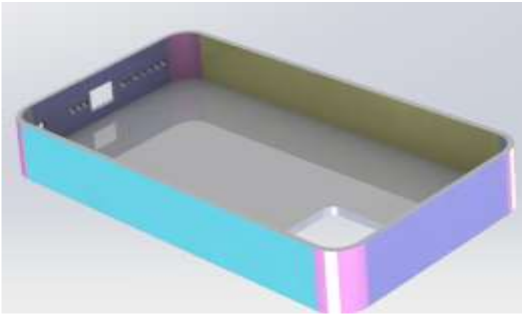
File Format

- .SLDPRT for SolidWorks parts
- .STL for 3D printing
- .STEP or .IGES for CAD data exchange
- .PDF for 2D drawings
- .DXF/DWG for 2D drawings and CNC
- .Parasolid (.X_T) for CAD compatibility



Application of learning 2.5.

As technician from computer peripheral manufacturing company you are requested to design the image below without considering scale and its size real size an export to STEP or .IGES for CAD data exchange.



Learning outcome 2 end assessment

Written assessment

Part I.Match the file format with its correct description:

1) .SLDPRT stands for the following:
a) A mesh format used for 3D printing
b) Native file format for parts in SolidWorks
c) Standard format for exchanging 2D drawings

2) . STL stands for:
a) A 2D format for sharing drawings
b) Converts 3D models into triangles, commonly used for 3D printing
c) A neutral format for CAD data exchange

3..STEP / .STP stands for:
a) Neutral format for sharing 3D CAD data across different platforms
b) Used for distributing technical drawings
c) Converts 3D models into mesh for printing

4..IGES / .IGS stands for:
a) Standard format for exchanging 2D drawings and technical designs
b) Used for transferring both 2D and 3D CAD data between platforms
c) Used for editing parametric models in SolidWorks

5).PDF stands for:

a) A 2D format for sharing technical drawings
b) A neutral format for CAD data exchange
c) A format used in CNC machining

6).DXF / .DWG stands for:

a) A format used for exchanging precise geometric models
b) Neutral format used for surface models
c) Standard formats for exchanging 2D drawings

PART II

1. What does the "Extruded Boss" feature do in SolidWorks?

A) Removes material by extending a 2D sketch along a curved path
B) Projects a 2D sketch linearly to create a 3D part
C) Connects two sketches along a curved path to create a solid feature
D) Rotates a 2D sketch around an axis to create a solid feature

2. Which feature creates a solid by rotating a 2D sketch profile around an axis?

A) Swept Boss
B) Extruded Cut
C) Revolved Boss
D) Lofted Boss

3. What is the primary function of the "Chamfer" feature?

A) Rounds off sharp corners
B) Creates a beveled edge between two or more faces
C) Removes material from the outer surface of a solid
D) Copies a feature in a circular pattern

4. Which feature removes material by revolving a 2D sketch around an axis?

A) Revolved Cut
B) Revolved Boss
C) Extruded Boss
D) Lofted Cut

5. Which feature is used to hollow out a solid by removing material from the outer surface?

A) Fillet
B) Shell
C) Rib
D) Swept Boss

6. Which pattern feature arranges multiple instances of a feature in a straight line?

A) Circular Pattern
B) Mirror
C) Linear Pattern
D) Lofted Boss

7. Which of the following is NOT a standard plane in SolidWorks?

A) Front plane
B) Top plane
C) Left plane
D) Right plane
8. How can you create a custom plane in SolidWorks?

A) Use the "Sketch" tab and select "Plane"
B) Go to the "Features" tab, click "Reference Geometry," and select "Plane"
C) Select an edge and use the "Extrude" feature
D) Open the "Insert" tab and click on "Custom Plane"
9. What is the purpose of a custom plane in SolidWorks?

A) To define a new surface on which sketches and features can be created
B) To rotate an object in the assembly
C) To apply material properties to a part
D) To add dimensions to a model

10. Which of the following is NOT a valid reference for creating a custom plane in SolidWorks?

A) A face
B) A plane
C) A point
D) A dimension

11. When creating a custom plane, what can you use to define its orientation and position?
A) Only existing planes
B) Any reference entity such as a face, edge, or point
C) Only reference points
D) Coordinate system only
12. What step comes immediately after selecting "Plane" from the "Reference Geometry" menu when creating a custom plane?

A) Click "OK" to create the custom plane
B) Choose the reference entities to define the plane's location and orientation
C) Specify the plane's material properties
D) Define the plane's color
Part III.

Question1: What is the primary purpose of visualization in 3Dmodeling?
A) To create physical simulations

B) To assist in technical drawings
C) To help users understand how a product will look, behave, and interact with its environment
D) To calculate mass and dimensions

Question2: Which of the following best describes the purpose of assigning materials in 3Dmodeling?
A) To change the visual appearance only
B) To simulate physical properties like strength and density and affect visual appearance
C) To control the lighting setup
D) To create camera perspectives
Question3: What is the main difference between material and appearance in 3Dmodeling?
A) Material changes the physical properties, while appearance changes only the visual look
B) Appearance affects the lighting, and material affects transparency
C) Material affects the background, and appearance adjusts camera settings
D) Material is for animations, and appearance is for rendering
Question4: What is the main purpose of defining a scene and environment in 3D rendering?
A) To change the visual appearance of materials
B) To control how the part interacts with light and create realistic backgrounds
C) To adjust the dimensions of the part
D) To evaluate the mass of the object
Question5: Why is view orientation important in 3Dmodeling?
A) It controls how light interacts with the part
B) It adjusts the physical properties of the material
C) It positions the part in 3D space for better navigation and presentation
D) It changes the visual appearance of the part

Question6: Which of the following options refers to a display style that uses shading and edges to represent a model?
A) Wireframe
B) Shaded
C) Transparent
D) Texture-mapped
Question7: How does lighting affect a 3D rendering?
A) It controls the camera's perspective
B) It simulates light interaction, impacting brightness, shadows, and reflections
C) It alters the material properties
D) It changes the object's mass
Question8: What is the purpose of adjusting camera settings in a 3D scene?
A) To change material properties
B) To control the lighting setup

C) To create specific view angles and perspectives for presentations
D) To simulate physical properties
Question9: Which tool is used to assess the physical and visual properties of a 3D model?
A) Appearance
B) Scene setup
C) Evaluate
D) Display style

Practical assessment

As computer system technician you have been tasked with designing a custom motherboard tray for a high-end computer chassis. The tray will need mounting points for a variety of components and will be made of aluminium. You will create the part in SolidWorks, visualize it, and export it for both 3D printing and CNC machining.

References

Ji, P. (2011). *SolidWorks essentials*. Concord, Massachusetts: Ji Pengcheng https://www.scribd.com/doc/194529688/SolidWorks-Essentials-Ver-2011

Marketing, T. (2020, March 25). *Choosing between 2D and 3D site models for construction*. Take-Off Pros. https://www.takeoffpros.com/2020/03/25/using-2d-or-3d-for-your-site-models/

Learning Outcome 3: Assemble parts

Indicative contents

3.1. Importing components/inserting components.

3.2. Applying mates

3.3. Exporting assembly

Key Competencies for Learning Outcome 3: Assemble parts

Knowledge	Skills	Attitudes
• Description of components • Definition of mates based on their types • Identification of file format	• Importing components/inserting components • Making assembly from a part • Browsing components • Applying mates • Exporting Assembly	• Having Passion • Having Adaptability • Having Critical thinking capabilities • Having Creativity • Being Problem Solver • Being Decision Maker • Being a Team worker

Duration10 hrs

Learning outcome 3 objectives:

By the end of the learning outcome, the trainees will be able to:

1. Import correctly components as related to the assembly template.
2. Importing correctly Non-SolidWorks Parts (External Formats) In SolidWorks.
3. Differentiate types of mates related to the assembly template.
4. Identify different types of file format compatible with SolidWorks.
5. Make correct assembly from part template.
6. Export properly assembly from part template

Resources

Equipment	Tools	Materials
✔ Computer ✔ Projector	✔ Storage media devices ✔ SolidWorks software	✔ Electricity ✔ Internet connectivity

Indicative content 3.1:Importing components/inserting components.

Duration: 1 hrs

Theoretical Activity 3.1.1: Description on importing components.

Tasks:

1 : Refer to the introduction provide the answers on the following questions:

i. What do you understand by the assembly from a part in SolidWorks?
ii. Describe computer components that can be imported or inserted in SolidWorks?
iii. What do you understand by existing Part or Assembly in SolidWorks?
iv. What do you understand by the term browsing components in SolidWorks?

2: Provide the answer on papers/flipcharts.

3: Present your findings

4: Follow an expert view from trainer and ask clarification .

5: For further understanding read the **Key readings 3.1.1** in trainee manual

Key readings 3.1.1.: Description of Importing Components/Inserting Components

- **Definition**
  - ✔ **Importing Components** in SolidWorks refers to the process of bringing external 3D model files or part files into a SolidWorks assembly environment. This feature allows designers and engineers to integrate pre-existing parts or assemblies from various file formats into their current design project, facilitating collaboration and enhancing design efficiency.
- **Make assembly from a part:** If you have already created a part and want to make an assembly from it, you can use the "Make Assembly from Part" feature. This allows you to convert the part into an assembly and add other components to it.
- **Selection of Component Part /Assembly**

  In **SolidWorks**, the Selection of Component Part/Assembly refers to the process of choosing parts or assemblies to work with in a design project. SolidWorks allows you to create both **individual parts** and **assemblies** of multiple parts,

- **Browsing components**: When inserting components, you can browse for the desired components using the file browser. You can navigate through folders, search for specific file names, and preview the components before inserting them into the assembly. By importing and inserting components, you can build the assembly by bringing together the individual parts and creating the desired structure. This allows you to visualize the interaction and relationships between the components and ensure a proper fit and assembly.

Practical Activity 3.1.2: Importing components

Task:

1 :Refer to the introduction you are requested to perform the following task
As a technician you are asked to insert a designed mother board, RAM and Import any other components l to be assembled in SolidWorks?
2: Read the key reading 3.1.2 in trainee's manual

3: Pay close attention to how the trainer demonstrates each step for inserting components.

4: Follow along as the trainer demonstrates the steps for inserting components.

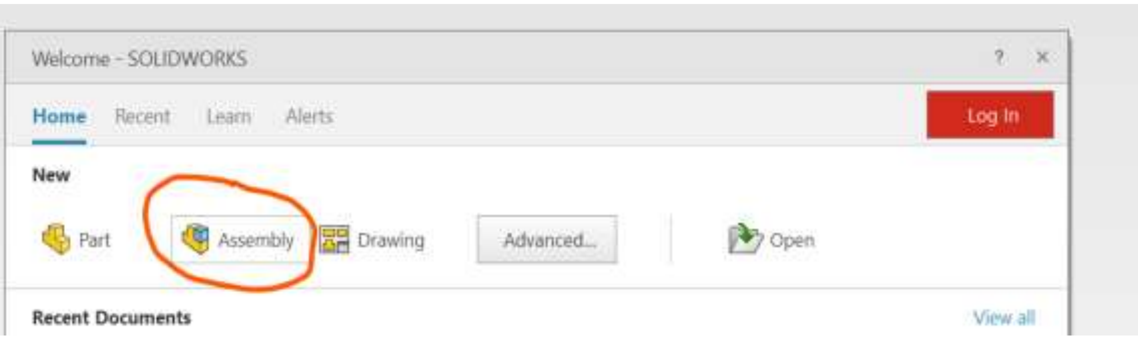
5: Ask for Clarification if any

Key readings 3.1.2: Importing Components/inserting Components

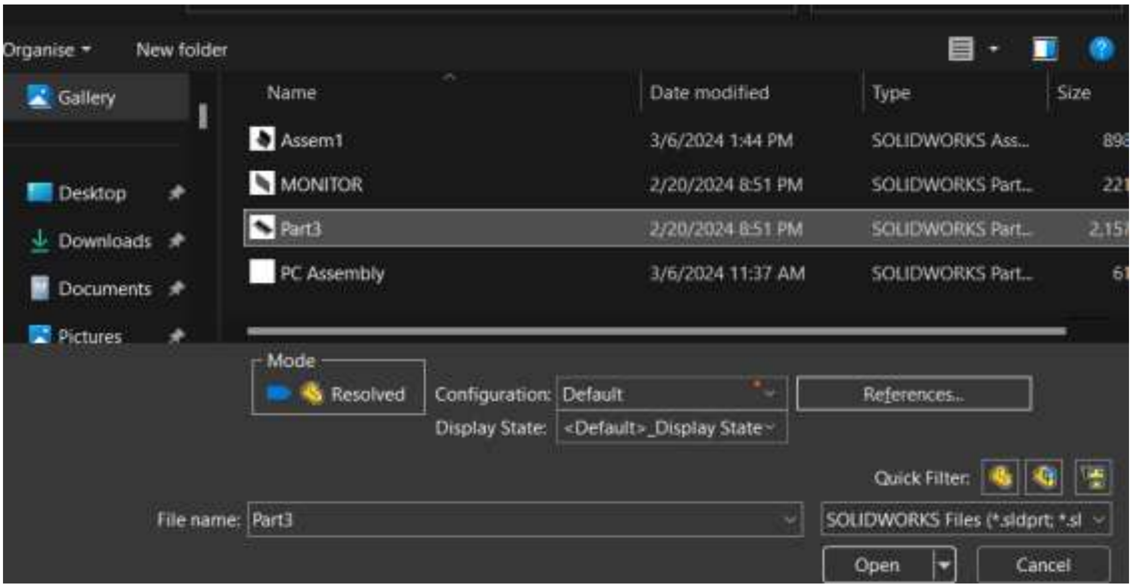
Steps for inserting components:

Step 1: Open SolidWorks

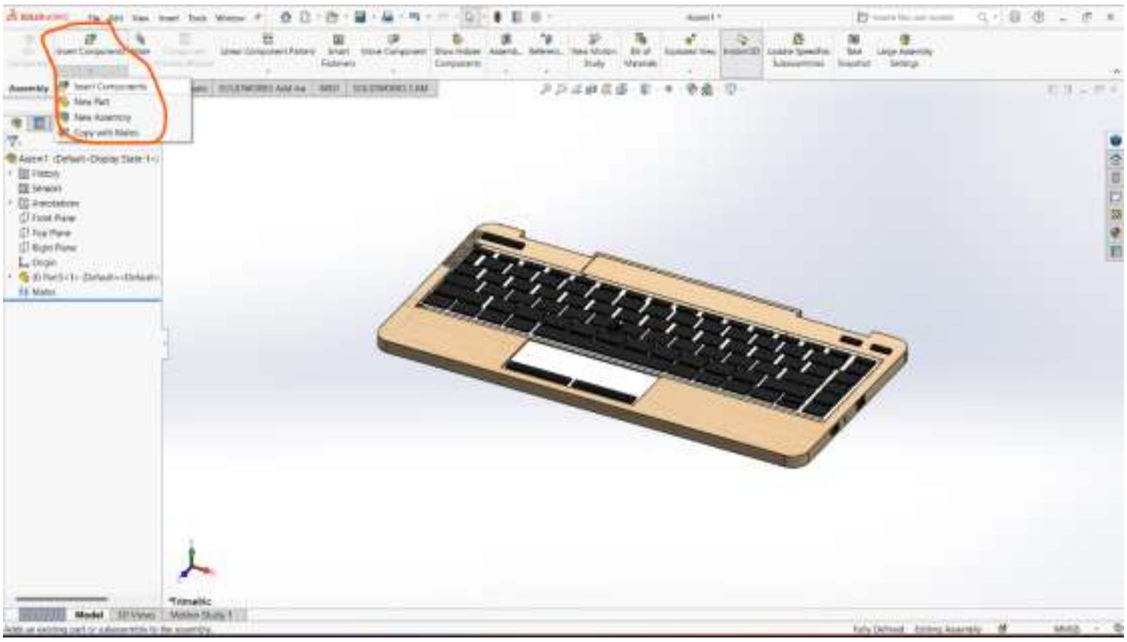
Step 2: Select assembly Template



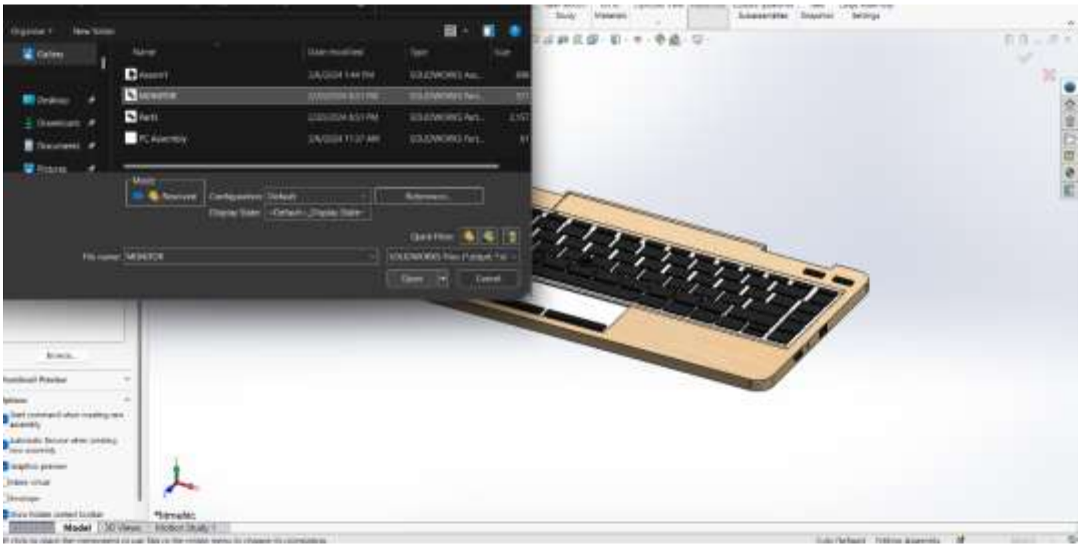
Step 3: Select first component/part from the location where it is saved then click **open**



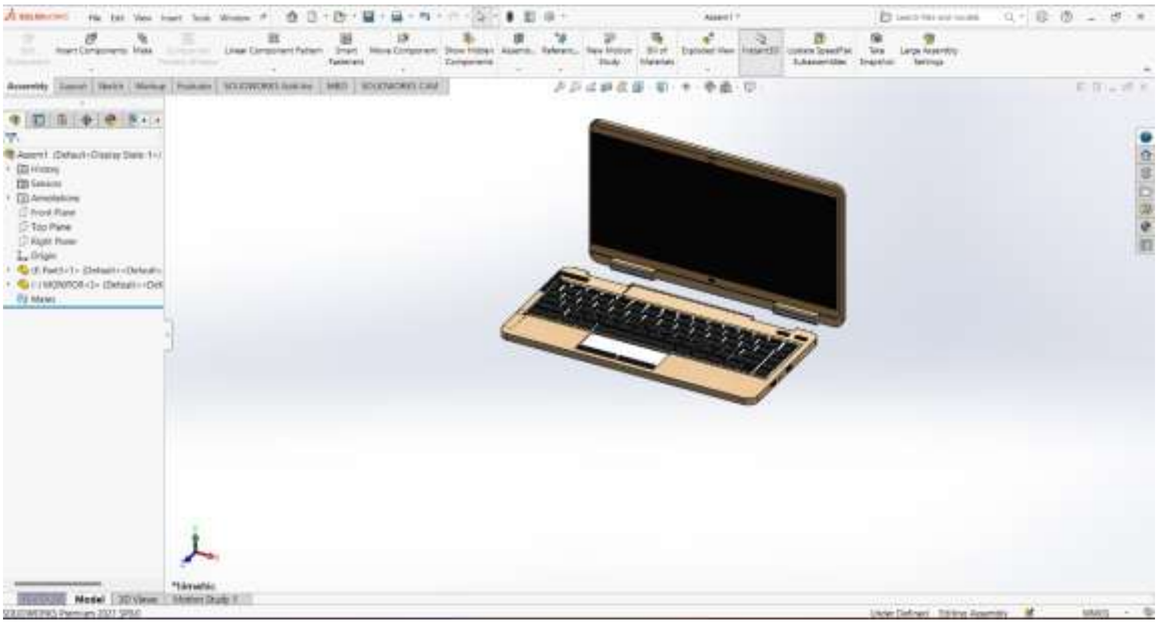
Step 4 : Open the Insert Component Tool and go in the Command Manager, click on the **Assembly** tab(to insert other parts/components).



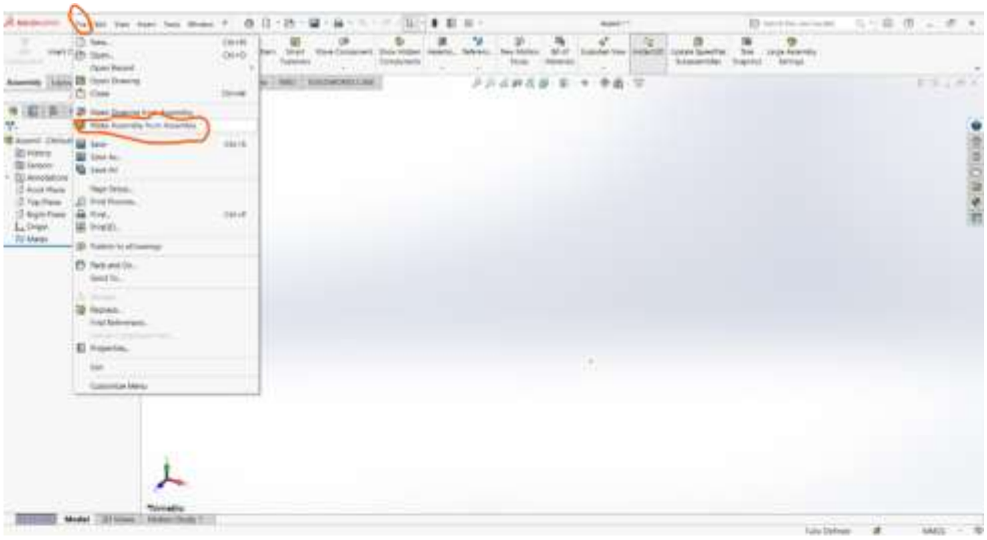
Step 5: Select **Insert Components** then, choose component that you want from its location and click on **Open**



Steps 6: Position your component/part



Step 7: Insert a Component from File go In the **Insert Components** window, click Browse and Navigate to the location where the part is saved (e.g., a previously created part or a part from another source) then Select the part file (SolidWorks Part file format, typically .SLDPRT) and after that click open .



Step 8 : Place the Component and After selecting the part/component, click anywhere in the workspace to place the component.

Importing Non-SolidWorks Parts (External Formats) in Solid works assembly template:

Importing External CAD Files you go to **File** > **Open**, and in the file type dropdown, select the appropriate file format (e.g., **STEP Files (.*STEP;*.STP)**).

- Import Options and click the Options button in the file dialog to configure import settings, such as whether to import features or treat the imported file as a "dumb" solid (a single, non-editable geometry).
- Insert into Assembly and once imported, you can insert the external component such like any other part by using the Insert Components tool in the Assembly.
- Save the Assembly and inserting and positioning your components, save your assembly by going to File > Save.
- Then Name your assembly and choose the desired save location.
- Apply positioning and mating Components

Points to Remember

✔ Trainee must know the steps of importing part components and non solid components.

Application of learning 3.1.

As technician electronic devices company, You are requested to make an assembly in SolidWorks for a new electronic device that includes both SolidWorks parts and external CAD components. Task is to import and place various components of the electronic device in the assembly workspace, ensuring proper placement of parts.

Indicative content 3.2: Applying mates

Duration: 8hrs

Theoretical Activity 3.2.1: Description of mates in SolidWorks

Tasks:

1 :Refer to the introduction you are asked to answer following questions:

- **i.** Describe the types of mates are used in SolidWorks?
- **ii.** Specify the purpose of each mate type as found in SolidWorks?
- **iii.** Identify the examples to each type of mate as found in SolidWorks?

2: Provide the answers on papers/flipcharts

3: Present your findings

4: Follow an expert view from trainer and ask clarification .

5: For further clarification ask question if any

Key readings 3.2.1.: Description of mates are used in SolidWorks

✔ **Definition and application of different types of mates:**

- In SolidWorks, mates are used to define relationships between components in an assembly, controlling how parts move relative to each other. They can be categorized into different types based on complexity and function:

1. Standard Mates

- Standard mates are the most basic and commonly used mates. They define simple geometric constraints between parts.

Types of Standard Mates:

- **Coincident:** Aligns two faces, edges, or points so that they lie on top of each other.
- **Parallel:** Forces two faces, edges, or planes to remain parallel to each other.
- **Perpendicular:** Makes two entities perpendicular to one another.
- **Distance:** Creates a fixed distance between two points, edges, or faces.
- **Angle:** Constrains two entities to remain at a specific angle.
- **Concentric:** Aligns two circular or cylindrical faces along a common axis.

- **Tangent:** Keeps two curved faces in contact at a single point, typically used with cylinders or spheres.

Standard mates are used for basic assembly tasks like fixing, rotating, or aligning parts.

2. Advanced Mates

- Advanced mates provide more complex control over part movement and relationships. They offer specialized constraints for precise mechanical design.

✔ **Types of Advanced Mates:**

- **Symmetric**: Keeps two components symmetric relative to a plane or axis.
- **Width:** Centers one component between two others, maintaining equal spacing on both sides.
- **Path Mate:** Constrains a component to move along a predefined path (e.g., a sketch).
- **Linear/Distance Limit:** Sets limits on how far two components can move relative to each other along an axis.
- **Angle Limit:** Constrains the rotation of components to within a specific range of angles.

✔ **Application:** Advanced mates are used when more control over movement is needed, such as limiting rotations or ensuring symmetry.

3. Mechanical Mates

- Mechanical mates simulate real-world mechanical relationships, such as gears or cam followers, allowing for realistic mechanical motion in assemblies.

✔ **Types of Mechanical Mates:**

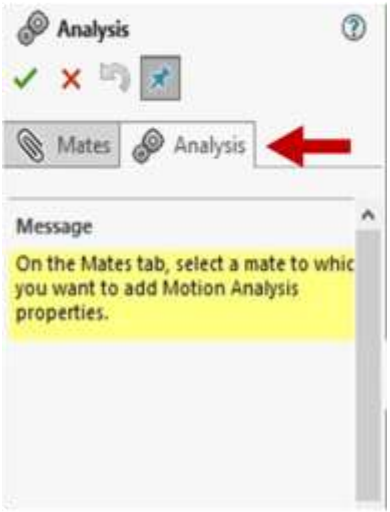
- **Cam:** Allows a follower component to move in contact with a cam component, simulating cam-follower motion.
- **Gear:** Simulates the relationship between two gears by ensuring that the rotation of one component drives the rotation of another.
- **Rack and Pinion:** Simulates the interaction between a linear component (rack) and a rotating component (pinion), converting rotational motion into linear motion.
- **Screw:** Constrains rotational motion and translates it into linear motion, simulating the behavior of a screw and nut.
- **Hinge:** Restricts motion to rotation around a single axis, simulating a hinge joint.
- **Slot:** Constrains one component to slide within a slot, mimicking slot-pin mechanisms.
- **Universal Joint**: Simulates the behavior of a universal joint, allowing two components to rotate relative to each other at variable angles.

✔ **Note that:** Mechanical mates are ideal for simulating specific mechanical interactions, making them useful for assemblies that have motion like gears, cams, and screws.

4. Mates for Analysis

- These mates are primarily used for motion analysis, where you want to evaluate how parts move and interact under certain conditions. While not technically a separate category, they play a key role in dynamic simulations.

  ✔ **Function in Analysis:**
  - Mates like Distance and Angle Limit are used in simulations to limit movement.
  - Mechanical mates are frequently used in kinematic analysis to simulate real-world interactions.
  - Path Mate and Linear/Angular Limit can also help study the motion and restrictions of mechanical systems.



- These mates are applied in motion studies or FEA (Finite Element Analysis) to analyse how parts behave under operational or stress conditions.

Practical Activity 3.2.2: Applying mates

Task:

1. Read the following task and perform the activity:

i. Refer to the details of the SolidWorks Parts like Laptop Base is a 300mm to 200mm as part component with slots for ports and ventilation, The Laptop Screen is a 300mm by 180mm display panel with a hinge mechanism that connects to the base. A task is to import all components both base and laptop screen then ensure that all laptop components are imported and assembled.

ii. Imagine you are designing a new telephone with two sections: a keypad and a touchpad.

Make sure to apply mates that allow the phone to fold and unfold while still functioning with one of its sections. Be sure to use both mechanical and standard mates to achieve this functionality.

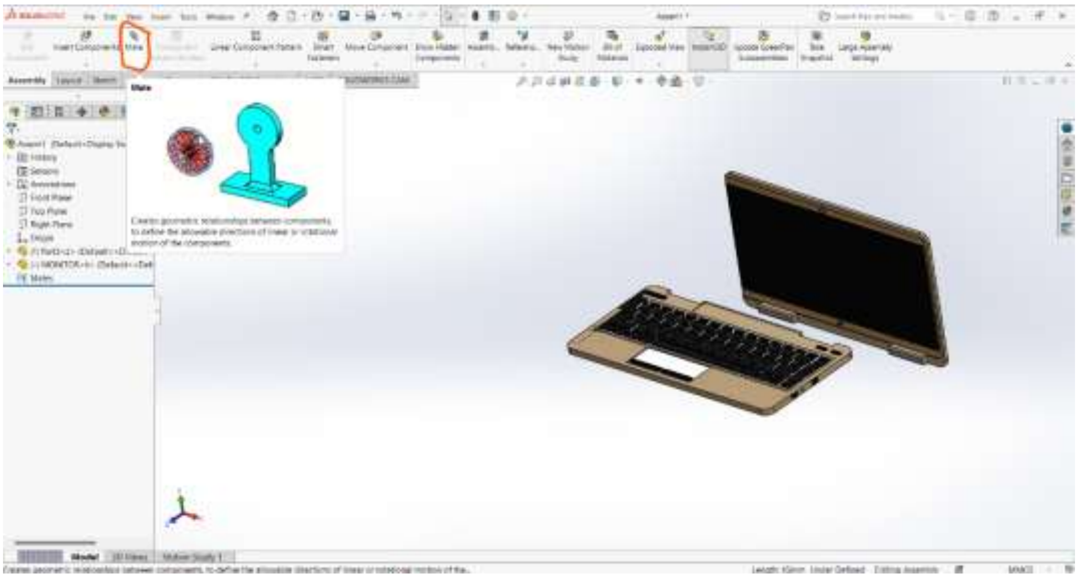
2. Read the key reading 3.3.2 in trainee's manual.
3. Pay close attention to how the trainer demonstrates each step for applying mates.
4. Follow along as the trainer demonstrates the steps for inserting components.
5. For further clarification, ask question if there is any.

Key readings 3.2.2: **Applying mates**

✔ **Steps to Apply Standard Mates in SolidWorks:**

Step 1: Open the Assembly

Step 2: Activate Mate Tool



Step 3: Select the Entities to Mate (face, edge, vertex, or axis) of the second component that you want to mate with the first one.

Step 4: Choose the Mate Type >Once both entities are selected, (Coincident, Parallel, Perpendicular and Specify a fixed distance and angle between the two selected entities.

Step 5: Adjust Mate Settings and alignment direction if necessary and preview to ensure it behaves as expected.

Step 6: Confirm the Mate

Step 7: Add More Mates (if needed) by Repeating

Step 8: Exit the Mate Tool

Note that:

- Once all required mates are applied, click the OK button to exit the Mate command.
- Test the assembly by dragging components to ensure that the mates behave as intended.
- **For example:** If you want to align two cylindrical parts:

1. Select the circular edge of the first part.
2. Select the circular edge of the second part.
3. Choose the Concentric mate.
4. SolidWorks will align the two parts along their axes.

Points to Remember

- **Types of mates**

1. Standard Mates
2. Advanced Mates
3. Mechanical Mates
4. Mates for Analysis

- You must know the steps to Apply Standard Mates in SolidWorks

Application of learning 3.2

As a technician in a computer peripheral manufacturing company, you are tasked with designing a new motherboard model. The motherboard will feature various components, such as a heat sink, processor, memory modules, capacitors, fan, and resistors. Ensure that each component is properly connected and aligned to create a functional assembly of the motherboard, with special focus on the heat sink and capacitors for optimal performance and cooling.

Indicative content 3.3: Exporting Assembly

Duration:1hrs

Theoretical Activity 3.3.1: Description on exporting an Assembly in SolidWorks

Tasks:

1. After forming small groups, answer the following questions:
   i. In SolidWorks, when exporting an assembly or part, why is it important to choose the correct file format ?
   ii. How does selecting the appropriate file location impact the workflow in terms of collaboration, version control, and accessibility?
2. Provide the answers on paper/flipchart.
3. Present your findings.
4. Follow an expert view from trainer and ask clarification.
5. For further clarification, Read the **Key readings 3.3.1** in trainee manual.

Key readings 3.3.1.:Exporting an Assembly in SolidWorks

- **Exporting assembly**

✔ **Exporting an assembly** refers to the process of saving a complete assembly file, which includes all its individual components and their relationships, into a different file format suitable for use in other software or for specific applications. This process allows the assembly to be shared, modified, or utilized in various contexts, such as manufacturing, 3D printing, or collaboration with other design tools.

✔ **File format:** SolidWorks offers various file formats to export assemblies. An assembly is a collection of related parts saved in one SOLIDWORKS document file with ".sldasm" extension. The other common file formats for exporting assemblies include:

  - **STEP (.stp):** This is a widely used file format for exchanging 3D models between different CAD software. It provides a neutral format that preserves the geometry and structure of the assembly.
  - **IGES (.igs):** IGES is another standard file format for exchanging 3D models. It allows for the transfer of geometry information between different CAD systems.
  - **Parasolid (.x_t or .x_b):** Parasolid is a 3D modeling kernel used in many CAD systems.

- **STL (.stl):** STL is a file format commonly used for 3D printing. Exporting an assembly in STL format creates a mesh representation of the assembly suitable for 3D printing applications.
- **PDF (.pdf):** If you need to export the assembly for documentation or viewing purposes, you can choose to export it as a PDF file.

✔ **File location:** When exporting an assembly, you need to specify the file location where you want to save the exported file. Choose a location on your computer or network where you can easily access the file later.

✔ **Know that**: Selecting the right file location in SolidWorks greatly affects workflow by enhancing collaboration, version control, and accessibility.

- **Collaboration:** Shared or cloud-based storage allows all team members to access the latest version of files, improving teamwork and reducing the chance of using outdated information.
- **Version Control:** Organizing files in a structured way helps track design progress and manage different versions, making it easy to revert to previous designs if necessary.
- **Accessibility:** Centralized storage ensures that authorized users can access files anytime and from any device, facilitating remote work and reducing delays in locating documents.

Practical Activity 3.3.2: Exporting an Assembly in SolidWorks

Task:

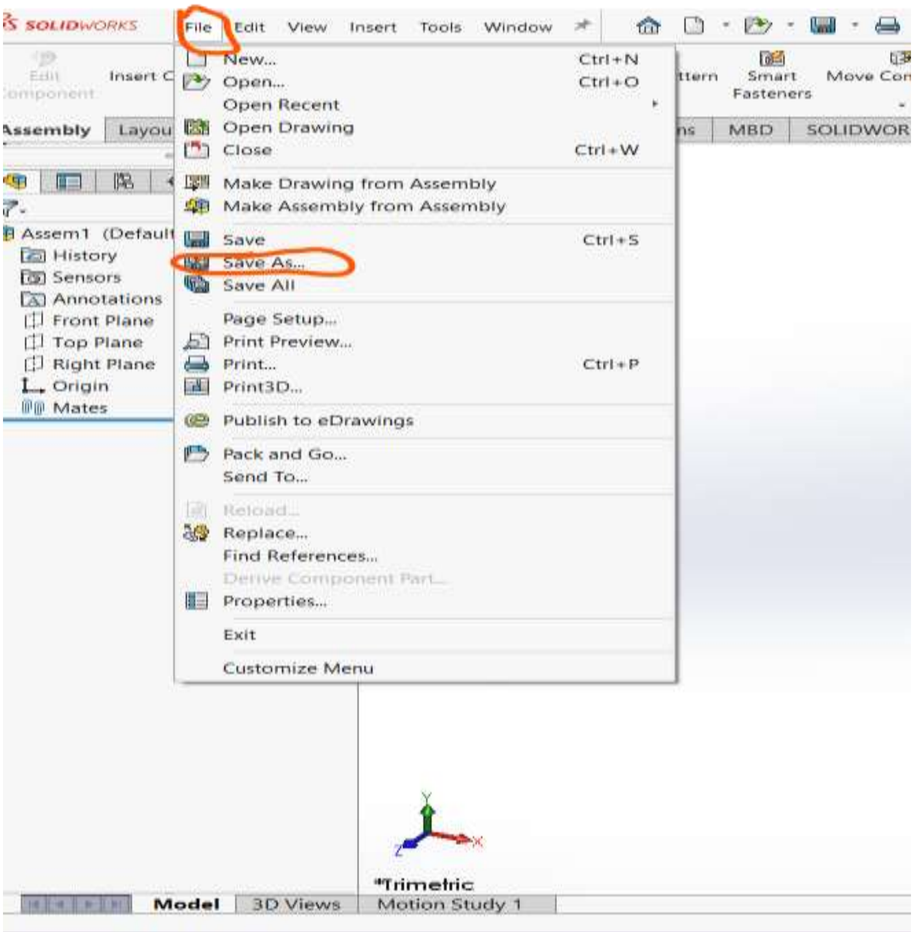
1. Refer to the introduction you are requested to perform the following task :
   In this activity, suppose that you have a model and assemble of laptop docking station in SolidWorks with components such as a docking base, USB and HDMI ports, an circuit board, and a power adapter and you are asked to use that model and make an assemble from the parts by mating the ports to the base, positioning the circuit board, and attaching the power adapter. Once verified, export the assembly in formats like STEP or STL for further use.
2. Read the **Key readings 3.3.2** in trainee manual
3. Learn how to export an assembly in SolidWorks from the trainer.
4. Follow along the steps of exporting Assembly in SolidWorks.
5. For more clarifications, ask question if any.

Key readings 3.3.2:Exporting an Assembly in SolidWorks

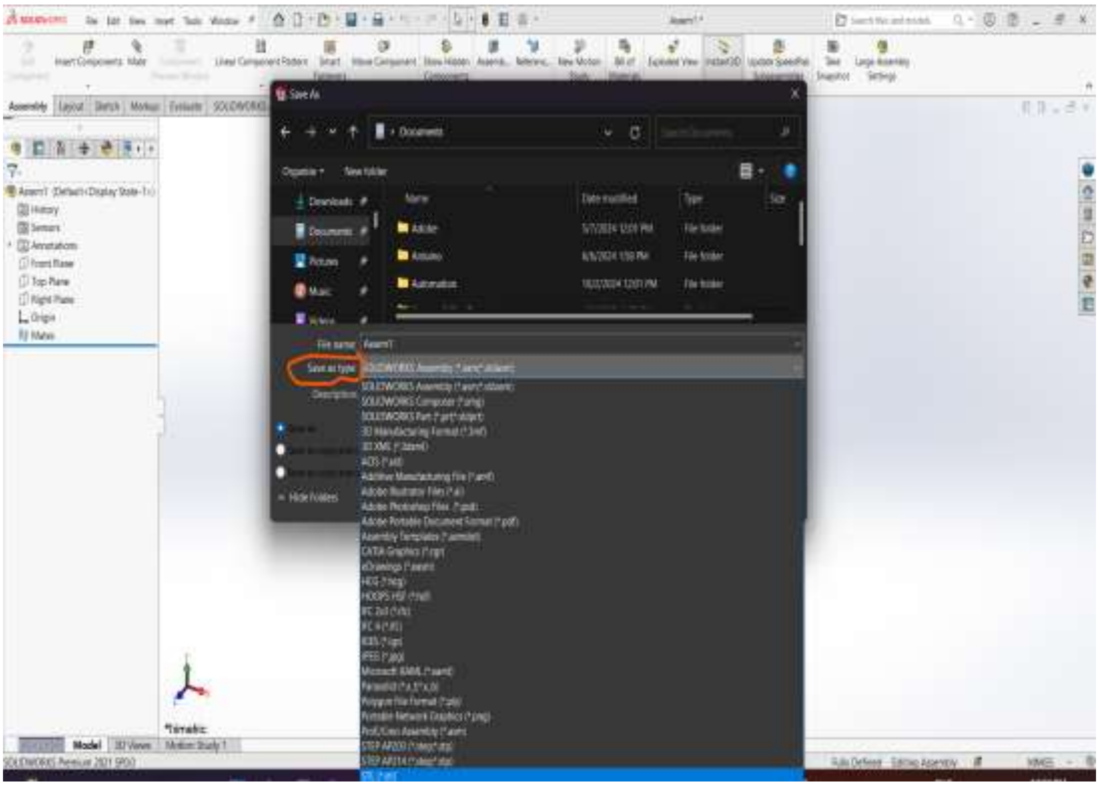
- **File location:** When exporting an assembly, you need to specify the file location where you want to save the exported file. Choose a location on your computer or network where you can easily access the file later.
- **File format:** SolidWorks offers various file formats to export assemblies. An assembly is a collection of related parts saved in one SOLIDWORKS document file with ".sldasm" extension. By exporting the assembly in the desired file format, you can share, collaborate, or use the assembly in other applications as needed.

✔ **Steps to be followed to export a file:**

1. Open Assembly:
2. Prepare the Assembly for Export
3. Access the Export Function by click on File, click on Save as.

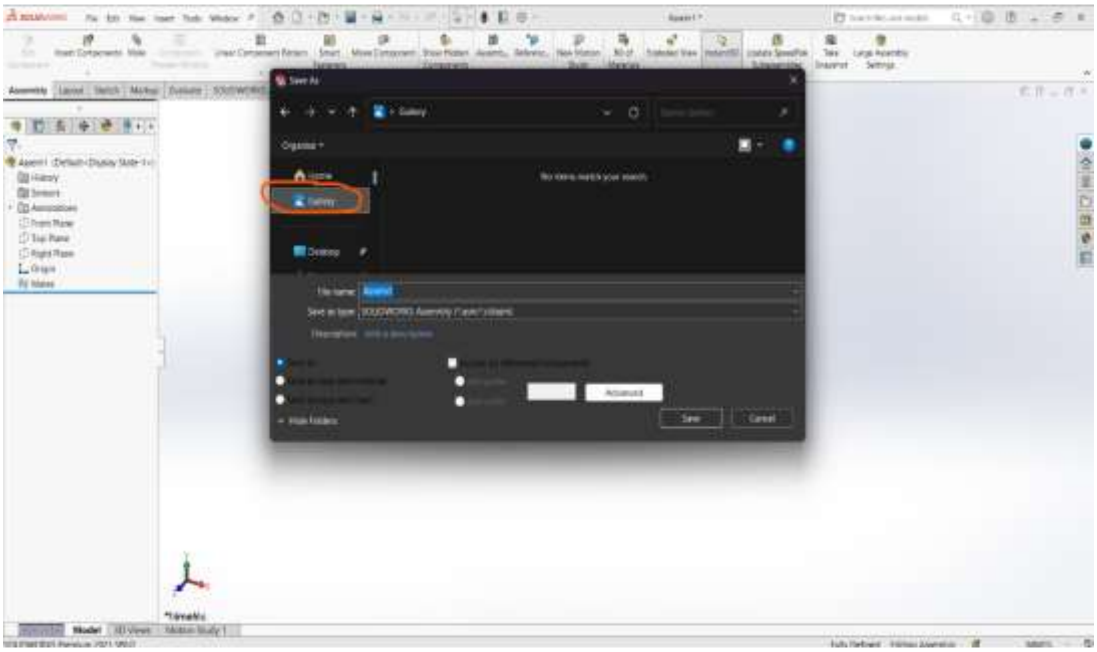


4. Select Export File Format



5.Configure Export Options

6.Choose the file location



7.Export the Assembly

8.Save and Close the Assembly

✔ **Note that:** After saving, you can close the assembly or continue working as needed

Points to Remember

✔ **File Format Options:**

- STEP (.stp
- IGES (.igs
- Parasolid
- STL (.stl)
- PDF (.pdf).

- It is necessary to choose File Location or file directory while exporting an assembly in SolidWorks.
- You must know the Steps for Exporting Assembly

Application of learning 3.3

HP provides various docking stations and monitors with standard dimensions to enhance user connectivity. As a technician, you will design a laptop and docking station in SolidWorks, using the specifications for the HP USB-C Dock G5 (300mm x 100mm x 10mm) and the HP Elite Docking Station (310 x 94 x 33 mm). Many HP monitors adhere to VESA mounting standards (100mm x 100mm or 75mm x 75mm) for compatibility. After completing the design, you must export the file, at least three different formats for review and further use.

Learning outcome 3 end assessment

Written assessment

Select the right answer:

1. **What feature allows you to create an assembly from an existing part in SolidWorks?**
   a) Insert Components
   b) Make Assembly from Part
   c) Assembly Wizard
   d) Build Assembly
2. **How can you add an existing part or assembly into a current assembly?**
   a) Use "Open" and select the part
   b) Click "Component" then select "Existing Part/Assembly"
   c) Drag and drop from the toolbar
   d) Use "Sketch" and create the part from scratch
3. **Which tool is used to browse for components to insert into an assembly?**
   a) Search Components
   b) Insert Components
   c) Component Library
   d) Browse Parts
4. **Where do you find the 'Insert Components' option in SolidWorks?**
   a) File Menu
   b) Tools Menu
   c) Assembly Tab in the Command Manager
   d) View Tab
5. **What file format is used when importing a SolidWorks part?**
   a) .STEP
   b) .SLDPRT
   c) .IGES
   d) .DXF
6. **To import a non-SolidWorks part like a STEP file, what must you do first?**
   a) Convert it in SolidWorks
   b) Use the 'Open' command and choose the correct file format
   c) Save it as a .SLDASM
   d) Use the 'Import Wizard'
7. **What is the purpose of applying mates in an assembly**
   a) To check interference
   b) To constrain components relative to each other
   c) To define material properties
   d) To export the assembly
8. **Which of the following is a standard mate type in SolidWorks?**
   a) Symmetry
   b) Tangent

c) Velocity
d) Contact

9. **What does the "Coincident" mate do?**
a) Aligns two components parallel to each other
b) Fixes components in one place
c) Ensures two faces or edges touch each other
d) Aligns components perpendicularly

10. **Which of the following is an advanced mate type?**
a) Distance
b) Angle
c) Width
d) Parallel

11. **What is the function of a mechanical mate in SolidWorks**
a) It constrains parts to rotate together
b) It simulates specific mechanical movements like gears or cams
c) It locks parts in place
d) It connects parts with fixed distances

12. **Which mechanical mate would you use to simulate a gear mechanism?**
a) Width Mate
b) Screw Mate
c) Gear Mate
d) Symmetric Mate

13. **What is the function of the "Distance" mate in SolidWorks?**
a) It fixes two components at a specific distance from each other
b) It aligns two components parallel to each other
c) It aligns two components perpendicularly
d) It sets two components to be coincident

14. **What type of analysis can you perform after applying mates?**
a) Force Analysis
b) Structural Analysis
c) Motion Analysis
d) Color Analysis

15. **What is the purpose of performing mate analysis in SolidWorks?**
a) To check if the mates cause interference between components
b) To verify mate positions in exploded views
c) To add more mates to an assembly
d) To remove redundant components

16. **When exporting an assembly, which command should you use to save the assembly?**
a) Export
b) Save As
c) Close and Save
d) Transfer File

17. **Which file format would you choose to export an assembly for use in another CAD software?**
    a) .SLDASM
    b) .STEP
    c) .TXT
    d) .XLS
18. **Where do you specify the location to save an exported assembly file?**
    a) In the Mates menu
    b) In the Save As dialog box
    c) In the Display tab
    d) In the Assembly Manager
19. **Which file format would you use to export a 2D drawing of your assembly?**
    a) .DXF
    b) .SLDPRT
    c) .IGES
    d) .PDF
20. **What should you do if you want to export the assembly including all associated components?**
    a) Select 'Save As' and choose 'Include All Files'
    b) Use 'Pack and Go' feature
    c) Compress the files manually
    d) Use the 'Assembly Wizard'

Practical assessment

As a technician at a computer peripheral manufacturing company tasked with designing and assembling components for a new device, including a simple motherboard for an electronic device. Your responsibilities include creating a SolidWorks assembly of the motherboard, inserting and positioning components such as the processor, memory modules, heat sink, capacitors, fan, and resistors. You need to apply appropriate mates to ensure proper alignment and connectivity between the components. Additionally, you will insert external components to create non-editable geometry as part of the assembly.Once the assembly is complete, you will prepare it for a client review by exporting it in multiple formats. This includes generating a PDF for documentation, a DXF file for integration with the client's existing CAD system, and a high-resolution image for presentation. It is essential to ensure the accuracy and suitability of the exported files for review and further use.

References

https://www.javelin-tech.com/blog/2021/02/how-to-install-solidworks-software/

SolidWorks 2024 for Designers by Prof. Sham Tickoo ISBN: 978-1-64082-168-6

Engineering Design with SolidWorks 2024" by David C. K. Hsu ISBN: 978-1-305-93944-7

SolidWorks 2024: A Power Guide for Beginners and Intermediate Users by Sandeep Dogra ISBN: 978-1-122-00448-6

Learning Outcome 4:Produce drawing

Indicative contents
Indicative content 4.1: Setting drawing sheet layout **Indicative content 4.2**: Editing the title block **Indicative content 4.3:** Importing views **Indicative content 4.4:** Applying Visualization **Indicative content 4.5:** Exporting the drawing

Key Competencies for Learning Outcome 4: Produce drawing

Knowledge	Skills	Attitudes
• Identification of drawing sheet layout standard size. • Identification the printer paper orientation	• Setting drawing sheet layout • Editing the title block • Importing views • Applying Visualization • Exporting the drawing	• Having Adaptability • Having Critical thinking capabilities • Having Creativity • Being Problem Solver • Being Decision Maker • Being a Team worker

Duration: 20 hrs

Learning outcome 4 objectives:

By the end of the learning outcome, the trainees will be able to:

1. Select properly a drawing sheet layout used to produce SolidWorks drawings.
2. Edit correctly the drawing sheet title block for producing SolidWorks drawings.
3. Import correctly different components and their views in SolidWorks
4. Apply proper visualization on drawing in SolidWorks software.
5. Export properly the drawing in different file formats.

Resources

Equipment	Tools	Materials
● Computer ● Projector	● SolidWorks software ● Storage nedia devices (Flash disk, External hard disk)	● Electricity ● Didactic materials

Indicative content 4.1: Setting drawing sheet layout

Duration: 3 hrs

Theoretical Activity 4.1.1: Identification of drawing sheet layout setting

Tasks:

1: Read carefully and answer the following questions

i. What are the standard sizes of drawing sheet?
ii. Discuss on two main orientations of drawing sheets, and when would each be used?

2: Write the findings on paper / flipchart.

3: Present the findings.
4: For more clarification, read key **readings 4.1.1** in train manuaul
5: In addition, ask questions where necessary.

Key readings 4.1.1: Setting drawing sheet layout

When creating a drawing in SolidWorks, you need to set the drawing sheet layout, including the sheet size and orientation. This determines the dimensions and orientation of the drawing sheet on which you will create your technical drawings. Here are the options for setting the drawing sheet layout:

✔ **Standard Sheet Sizes in SolidWorks**

SolidWorks provides several standard sheet sizes to choose from when creating a drawing. These sizes follow international standards such as ISO (International Organization for Standardization) or ANSI (American National Standards Institute).

✔ **Standard ISO Sheet Sizes:**

- **A0**: 841 x 1189 mm
- **A1**: 594 x 841 mm
- **A2**: 420 x 594 mm
- **A3**: 297 x 420 mm
- **A4**: 210 x 297 mm

✔ **Standard ANSI Sheet Sizes:**

- **A** (Letter): 8.5 x 11 in
- **B**: 11 x 17 in

- **C**: 17 x 22 in
- **D**: 22 x 34 in
- **E**: 34 x 44 in

✔ **Sheet orientation**: The sheet orientation determines the alignment of the drawing sheet, whether it is in landscape (horizontal) or portrait (vertical)

✔ **Landscape orientation**: The sheet is wider than it is tall. This orientation is suitable for drawings that have a wider aspect ratio or require more horizontal space to accommodate the drawing views or dimensions.

✔ **Portrait orientation**: The sheet is taller than it is wide. This orientation is suitable for drawings that have a taller aspect ratio or require more vertical space to accommodate the drawing views or dimensions.

The choice of sheet orientation depends on the nature of your drawing and the space required to present the information effectively.

By setting the drawing sheet layout, including the sheet size and orientation, you can create a suitable canvas for your technical drawings in SolidWorks.

✔ **Custom Sheet Size:**

You can choose to create a custom sheet size if your project requires specific dimensions that are not available in the standard sizes. In this case, you can specify the width and height of the custom sheet size according to your needs.

✔ **To define Custom Size:**

To create a custom size, right-click on **Sheet1** and choose **Properties**.

Under **Sheet Format**, select **Custom Size** and enter the desired width and height.

Practical Activity 4.1.2: Setting Drawing Sheet Layout in SolidWorks

Tasks:

1. Refer to the introduction perform the activity of the following tasks:

   As a computer technician you are asked to choose and modify the drawing sheet orientation from landscape to portrait in SolidWorks?

2: Read **Key readings 4.1.2** in trainee manual

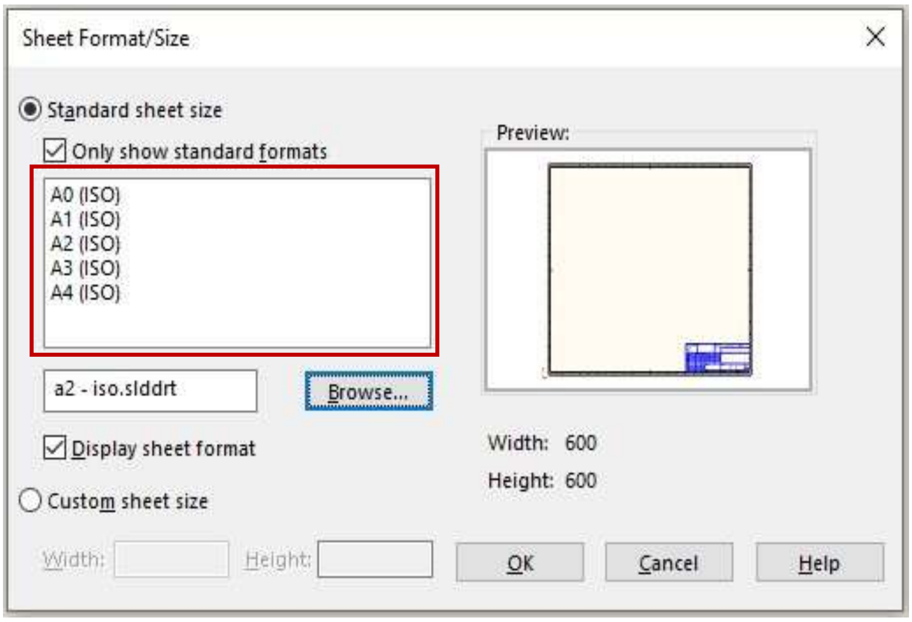
3: Learn to modify a SolidWorks drawing sheet layout from the trainer.

4: Modify drawing sheet orientation referring to the demonstrated steps of trainer

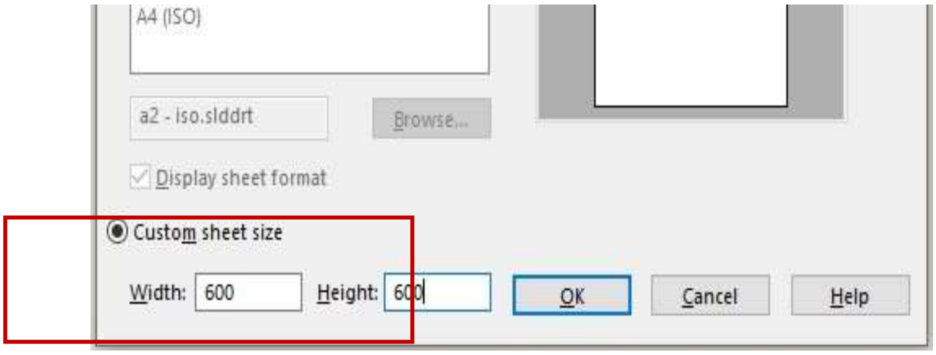
5:Ask questions for any further clarifications

Key readings 4.1.2 :Setting Drawing Sheet Layout in SolidWorks

- Start a New Drawing
- Select Sheet Format and Size



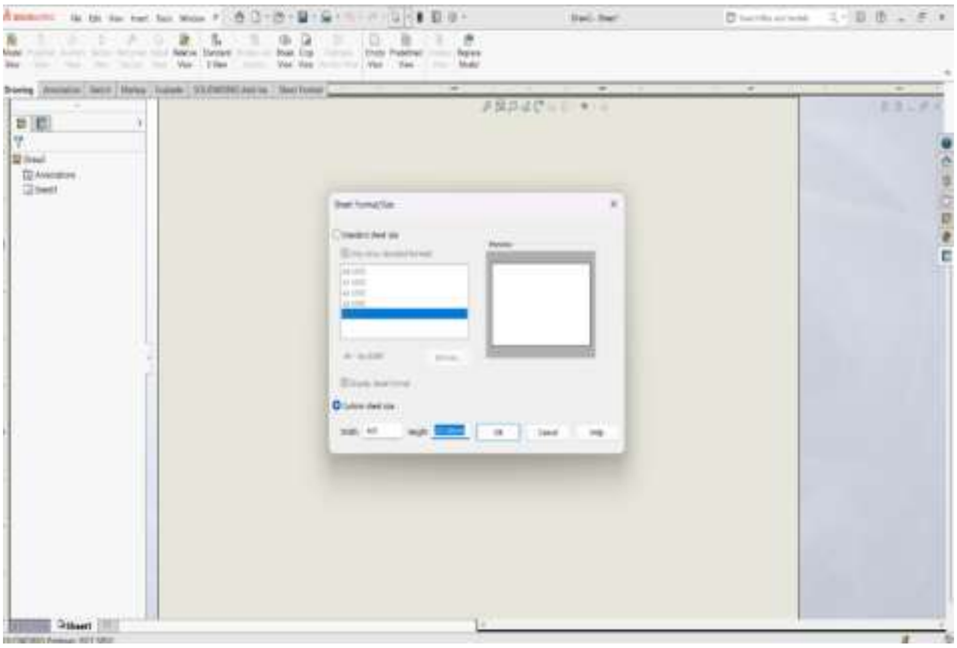
You can choose standard sheet size as shown on the figure above or customize the sheet size as shown on the figure below:



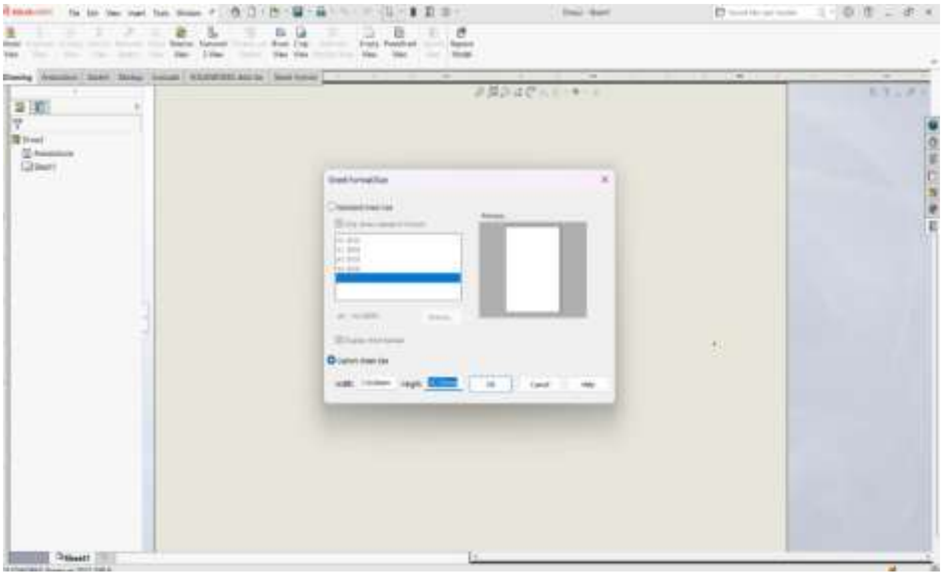
- **Set Up the Drawing Sheet Layout**

✔ **Sheet Orientation**:

- **Landscape**: Wider than tall; suitable for wider aspect ratio drawings.



- **Portrait**: Taller than wide; suitable for taller aspect ratio drawings.



✔ **Steps to Set Drawing Sheet Orientation**:

- **Open or Create a Drawing**:

Open an existing drawing or create a new one by navigating to File > New > Drawing.

- **Access Sheet Properties**:

In the Feature Manager Design Tree, right-click on Sheet1 (or the relevant sheet name). Select Properties from the context menu.

- **Set Sheet Size** (Optional):

Under **Sheet Format**, choose the desired sheet size from the standard options (ISO, ANSI) or define a **Custom Size** by entering width and height values.

Points to Remember

- Standard Sheet Sizes:

- ✔ ISO Sizes:
- ✔ ANSI Sizes:

- Trainees should know to customize Sheet Size
- Sheet orientation:
- Choosing between standard and custom sizes:

Application of learning 4.1

A 3D designing company needs a professional technician to provide and set a very well-organized drawing sheet layout for a 3D part designed in SolidWorks. As a technician who is familiar with SolidWorks design software you are hired to set the drawing sheet layout for presenting parts designed in SolidWork?

Indicative content 4.2: Editing the title block

Duration: 4 hrs

Theoretical Activity 4.2.1: Description of Title block

Tasks:

1: Read the following questions and provide the answers

i. What do you understand by the term title block?

ii. What essential information should be included in a title block, and how does it aid in identifying the drawing?

iii. Which of the following items is not typically included in a standard title block?

a) Author's name

b) Part number

c) Material specifications

d) Drawing scale

2: Provide the answer and write them on papers/ flip chart
3: Present the findings/answers
4: Pay attention to the expert view from trainer
5: In addition, ask questions where necessary For more clarification and read the key **readings 4.2.1.** in the trainee manual

Key readings 4.2.1.: Description of Title block

In SolidWorks, the title block is an important component of a drawing that contains essential information about the drawing, such as the title, author, date, part number, and revision history. Here are the options for editing the title block:

✔ **Title block table**:

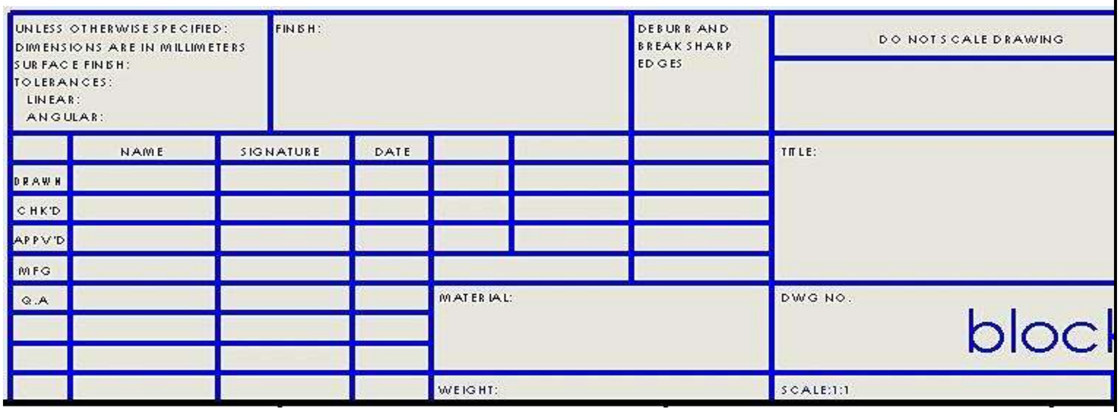
● **The title block** in SolidWorks is typically presented as a table format, allowing you to input and display information in a structured manner.

● **A title block** is a sheet template that includes a border and details about the design firm, project, client, and sheet, such as names, logos, addresses, issue dates, and revisions.

✔ **Title block information / elements of title block:** The title block in SolidWorks typically includes several elements of information that are commonly found in engineering drawings. These elements may vary depending on the specific requirements of your organization or project. Some common elements found in a title block include:

● **Drawing title**: The name or description of the drawing. Part number: The unique identifier for the part or assembly represented in the drawing.

- **Revision history**: The record of revisions made to the drawing, including revision letters, dates, and descriptions.
- **Author**: The name or initials of the person who created the drawing.
- **Date:** The date when the drawing was created or last modified.
- **Scale**: The scale at which the drawing is presented.
- **Company logo**: The logo or emblem of the company or organization associated with the drawing.



These elements can be customized and arranged within the title block table to meet your specific requirements.

By editing the title block in SolidWorks, you can ensure that the necessary information is accurately represented on your technical drawings and convey important details about the drawing to others.

Practical Activity 4.2.2: Editing title block

Task:

1: Refer to the introduction read and answer the following questions:

As a technician who have skills in SolidWorks,you are asked to set and edit the Title Block information in SolidWorks template and insert elements of title block

Read the Key readings 4.2.2 in trainee manual

3: Pay attention on demonstrates on how to edit title block in SolidWorks.

4: follow along the steps of editing title block in SolidWorks.

5: For more clarifications, ask question if any.

Key readings 4.2.2.Editing the title block

To edit the title block table, you can perform the following actions:

Add rows or columns: You can add additional rows or columns to the title block table if you need to include more information or customize the layout.

Merge cells: You can merge adjacent cells to create combined cells for specific information, such as merging cells to create a single cell for the drawing title or part number.

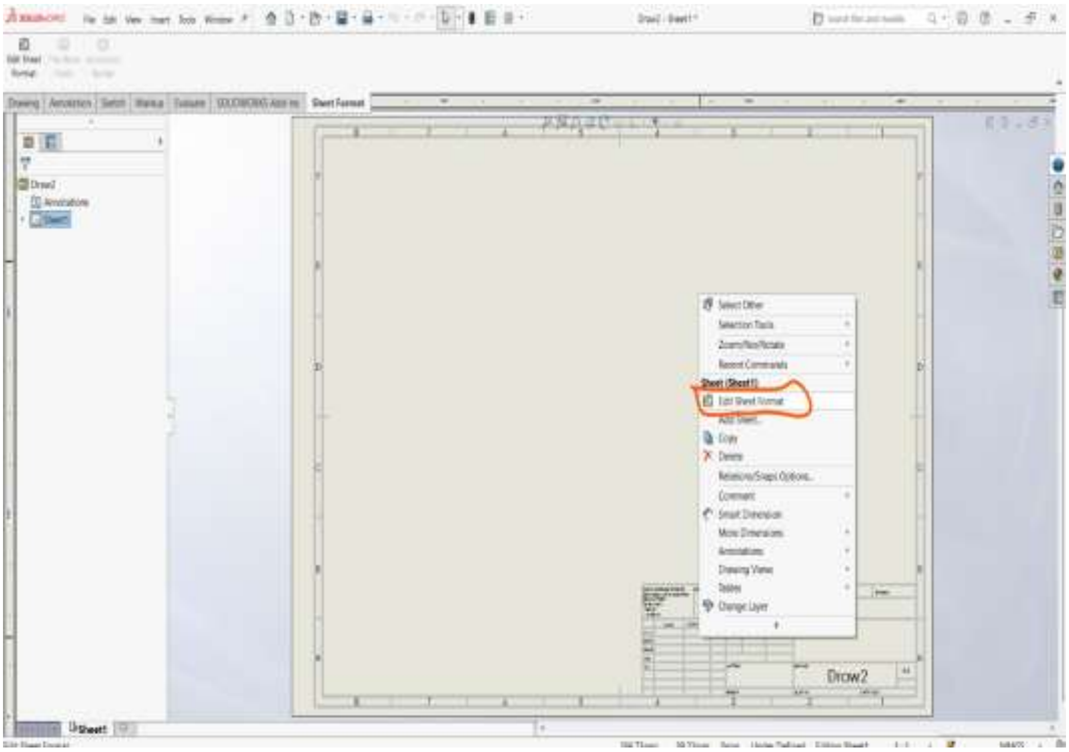
Split cells: You can split cells into multiple smaller cells if you want to divide information into separate sections within the title block.

Format cells: You can apply formatting options to the cells, such as changing the font, font size, alignment, and borders, to enhance the visual appearance of the title block.

Steps for editing title block in SolidWorks

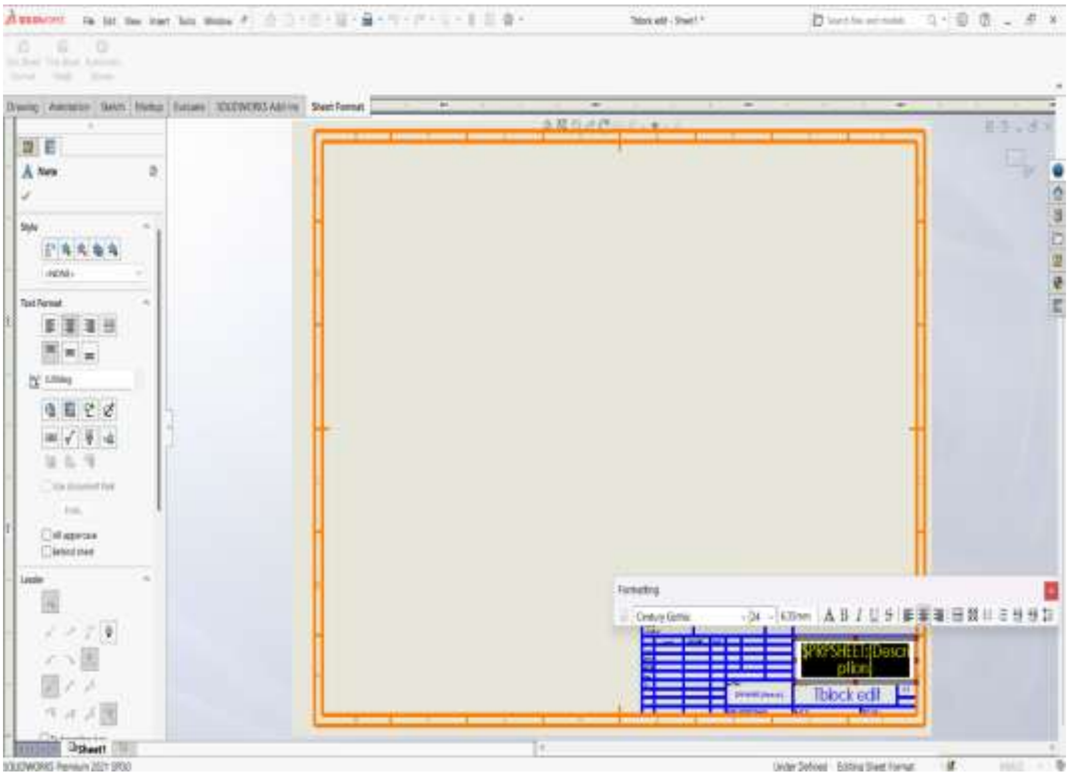
Open SolidWorks and select drawing template

To enter Edit Sheet Format Mode do Right-Click anywhere in the blank area of the drawing sheet then edit sheet format



Select a Text Field and Click directly on any of the text fields within the title block (such as the company name, drawing number, or date).

Click on the title block where you want to insert the new text field, and a cursor will appear for you to type the new information (e.g., revision number, designer's name).

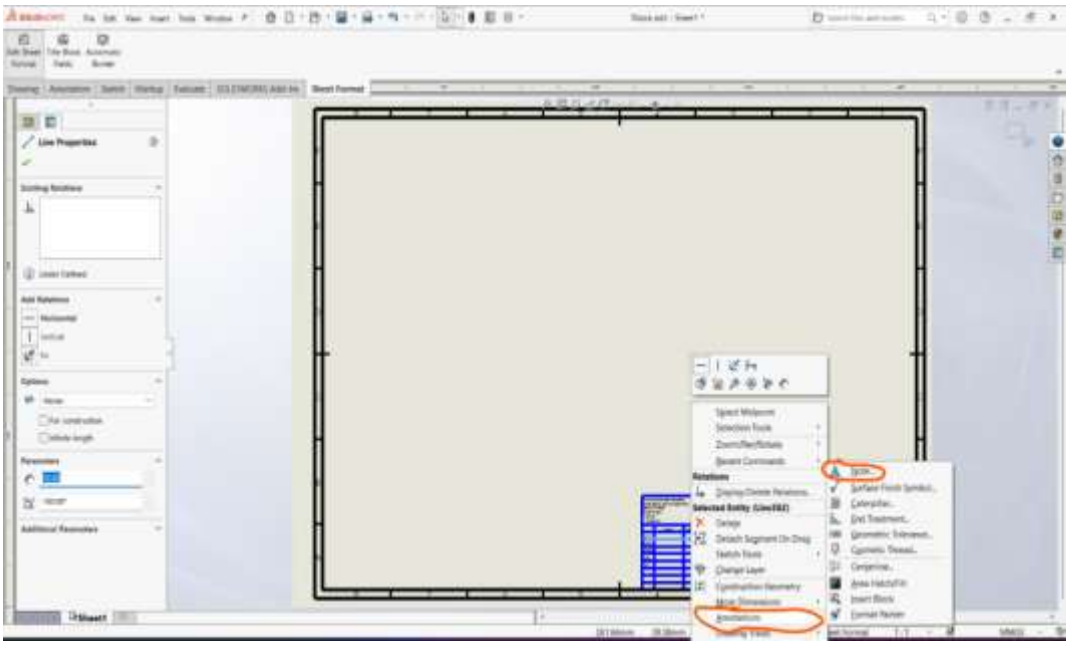


Insert a New Text Field or **If you want to add new text fields to the title block**:

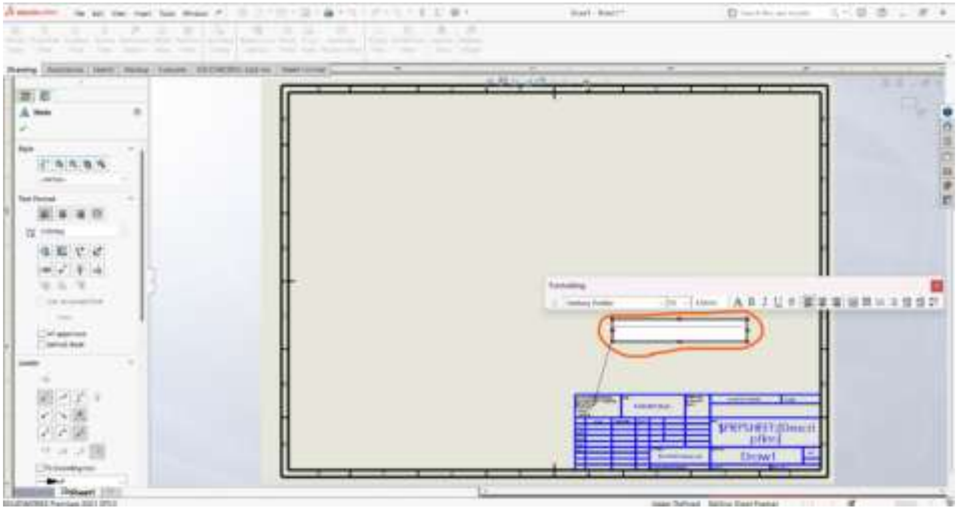
Right click

Go to the Annotation tab in the Command Manager

Click on Note.



Add Notes/texts that you want



Insert Custom Logos or Images (Optional) click on Insert > Picture> Browse and Select Image>Position and Resize the Image.

Go to **File** > **Save** to save the changes made to the drawing and Exit.

Points to Remember

- **The title block in SolidWorks is** a table format that organizes essential drawing information. You can customize it by adding, merging, splitting, and formatting cells.

✔ **Common elements in a title block include:**

  - Drawing title
  - Part number
  - Revision history
  - Author
  - Date
  - Scale
  - Company logo

- You must to edit Sheet Format Mode
- You must know the steps are used for editing existing text fields in title block
- Inserting new text fields to insert new text fields for additional information

Application of learning .4.2

You are working as a products designer in a company that manufactures Electronics components. Your company has secured a new project to design an electronic circuit **enclosure** for a major electronics client. As part of the project, you need to create detailed technical drawings in SolidWorks for the manufacturing team.

Before sharing the drawings with the client and the manufacturing team, you need to customize the Title Block on the drawing sheets to include specific information about the project, the client, and your company. The drawing must also adhere to the company's standards, include the client’s logo, and ensure proper documentation.

Indicative content 4.3: Importing views

Duration: 6 hrs

Theoretical Activity 4.3.1: Description on Importing Views in SolidWorks

Tasks:

1: Read carefully and provide answers to the following questions

i. Discuss on updating view palette in SolidWorks?

ii. What are the different types of views that can be imported into a drawing in SolidWorks?

2: Provide the answer for the asked questions and write them on papers.

3: Present your findings findings/answers.

4: For more clarification, read the key **readings 4.3.1.** in trainee manual

5: In addition, ask questions where necessary.

Key readings 4.3.1.: Description on Importing Views in SolidWorks

- **Importing views**

  In SolidWorks, importing views into a drawing allows you to display different perspectives and representations of your model. Here are the options for importing views:

- **Updating view palette**: The view palette in SolidWorks provides a visual representation of the available views that can be imported into your drawing. To update the view palette, you can perform the following actions:

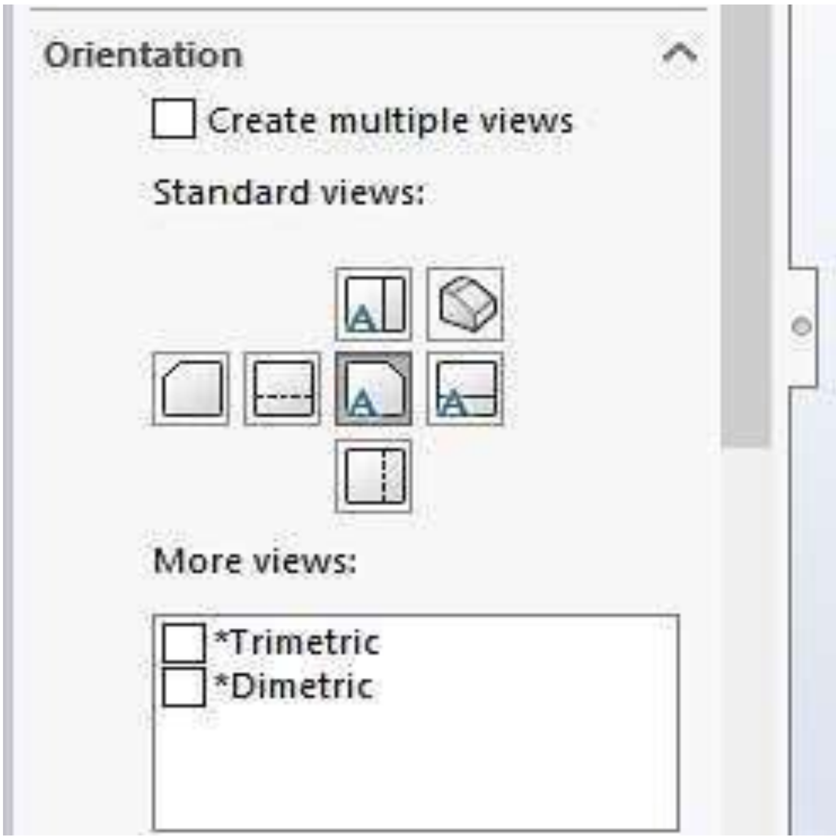
- **Refresh views**: If you have made changes to your model or added new views, you can refresh the view palette to update the list of available views.

- **Customize view palette**: You can customize the view palette by adding or removing views from the list, rearranging the order of views, or grouping views based on specific criteria.

✔ **Filter views**: You can apply filters to the view palette to narrow down the list of available views based on specific criteria, such as view type, orientation, or custom properties.

✔ **Model view**: A model view is a specific representation of your model that you can import into your drawing. Here are some options for creating and importing model views:

✔ **Standard 3D views**: SolidWorks provides standard 3D views, such as isometric, front, top, right, and so on, which can be easily imported into your drawing.



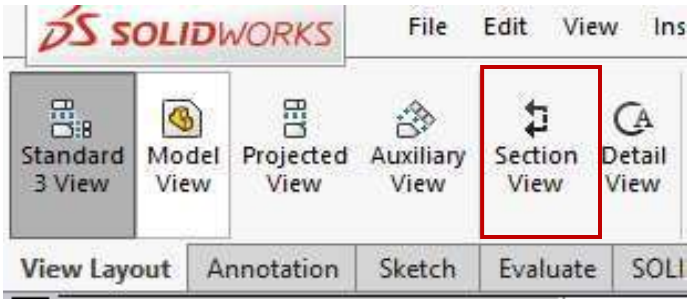
✔ **View Orientation**

- **Shortcut Keys:**
  - Ctrl + 1 = Front view
  - Ctrl + 2 = Back view
  - Ctrl + 3 = Left view
  - Ctrl + 4 = Right view
  - Ctrl + 5 = Top view
  - Ctrl + 6 = Bottom view
  - Ctrl + 7 = Isometric view

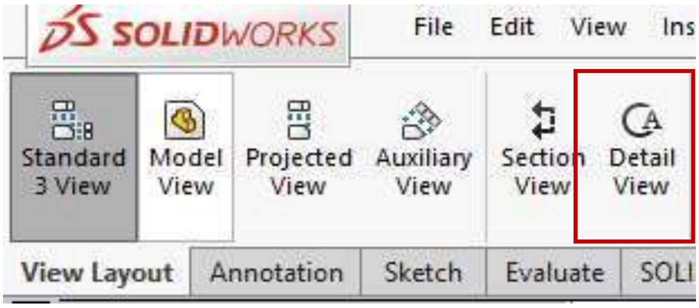
- Ctrl + 8 = Normal To selection

✔ **Custom 3D views**: You can create custom 3D views in SolidWorks by rotating, panning, or zooming in/out on your model to capture a specific perspective. These custom views can then be imported into your drawing.

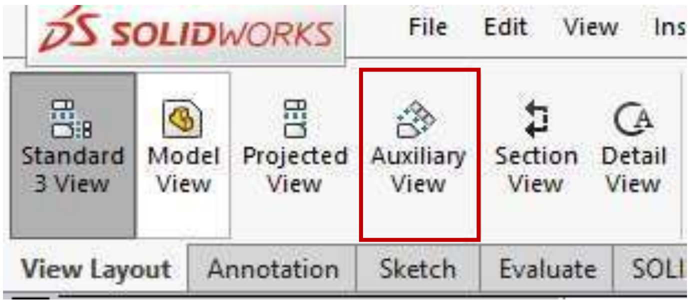
✔ **Section views**: Section views allow you to cut away a portion of your model to show internal features or hidden details. You can create section views and import them into your drawing.



✔ **Detail views**: Detail views are used to magnify a specific area or feature of your model. You can create detail views and import them into your drawing.



✔ **Auxiliary views**: Auxiliary views are used to show the model from a different angle or orientation to provide additional information. You can create auxiliary views and import them into your drawing.



✔ **Notethat:**By importing views into your drawing, you can present different perspectives, details, and sections of your model to communicate the necessary information effectively.

Practical Activity 4.3.2. Importing views

Task:

1: Refer to the introduction read and complete the following activity:

Suppose that you have a 3 D object designed in SolidWorks, you are asked to import a model views, align/position them into a new SolidWorks drawing.

2: Read Key readings **4.3.2 in Trainee Manual**

3: Pay attention to the trainer Demonstration on hwo to import view.

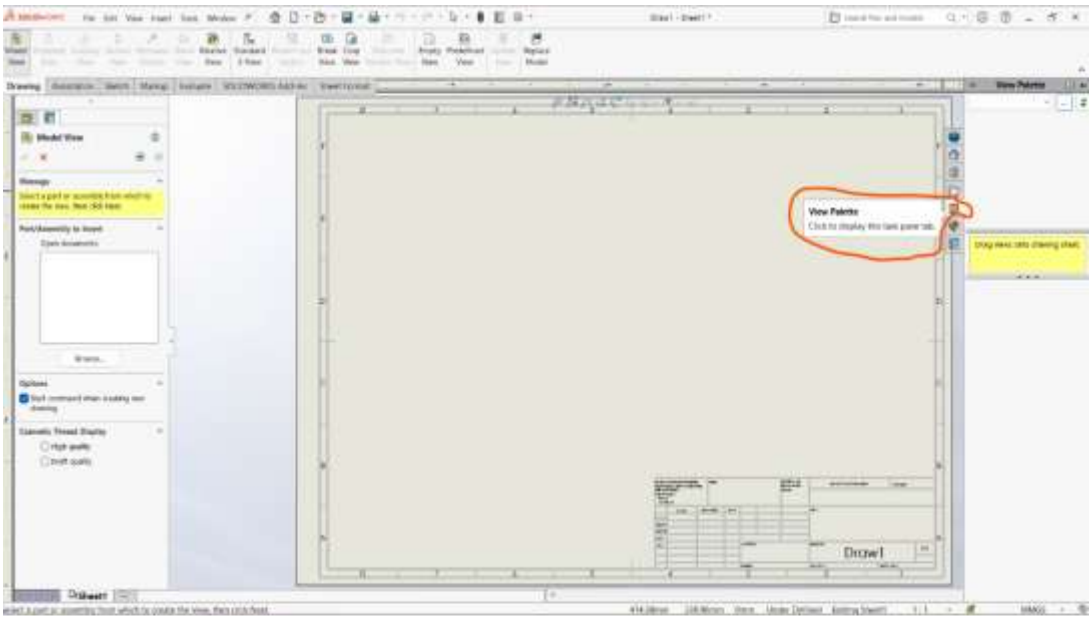
4: Follow along the steps are demonstrated by trainer then import view.

5: Ask for Clarification if any

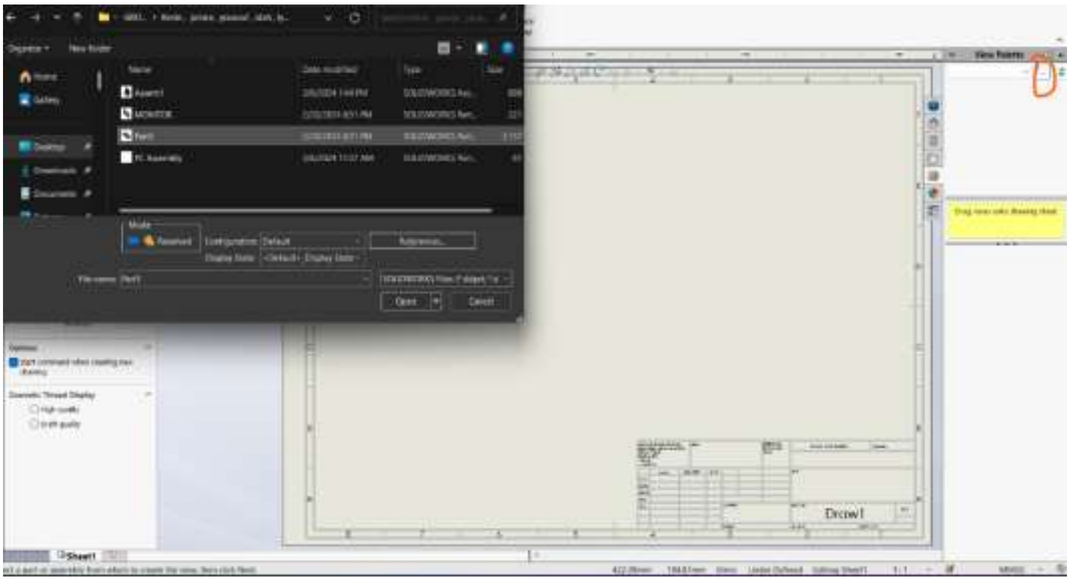
Key readings 4.3.2.: Importing Views in SolidWorks

In SolidWorks, importing views into a drawing allows you to display various perspectives of your model to effectively communicate information.

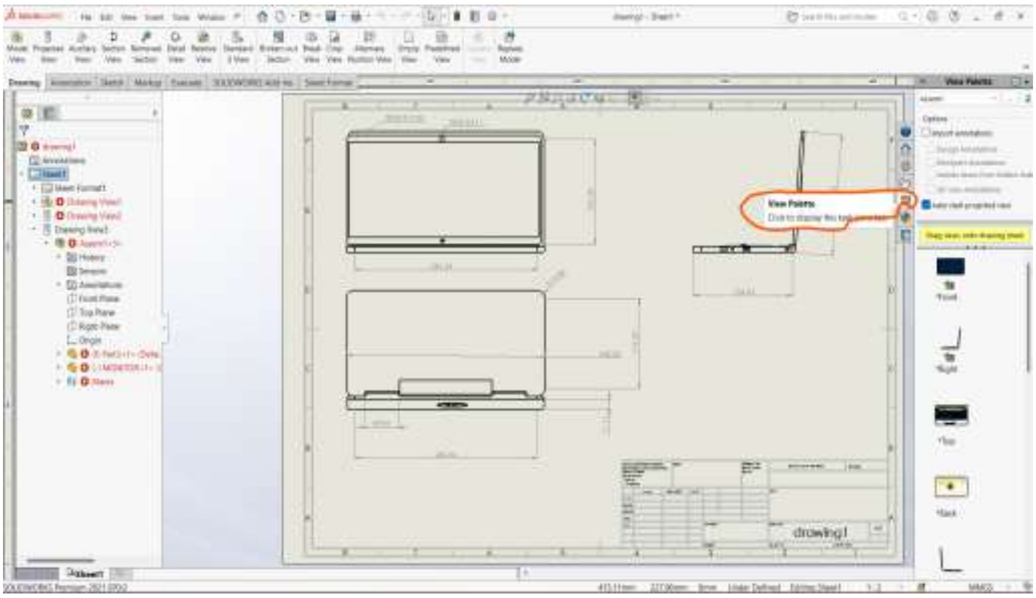
- **Updating View Palette**:
  - Open SolidWorks and choose **Drawing template**
  - Click on **View Palette**



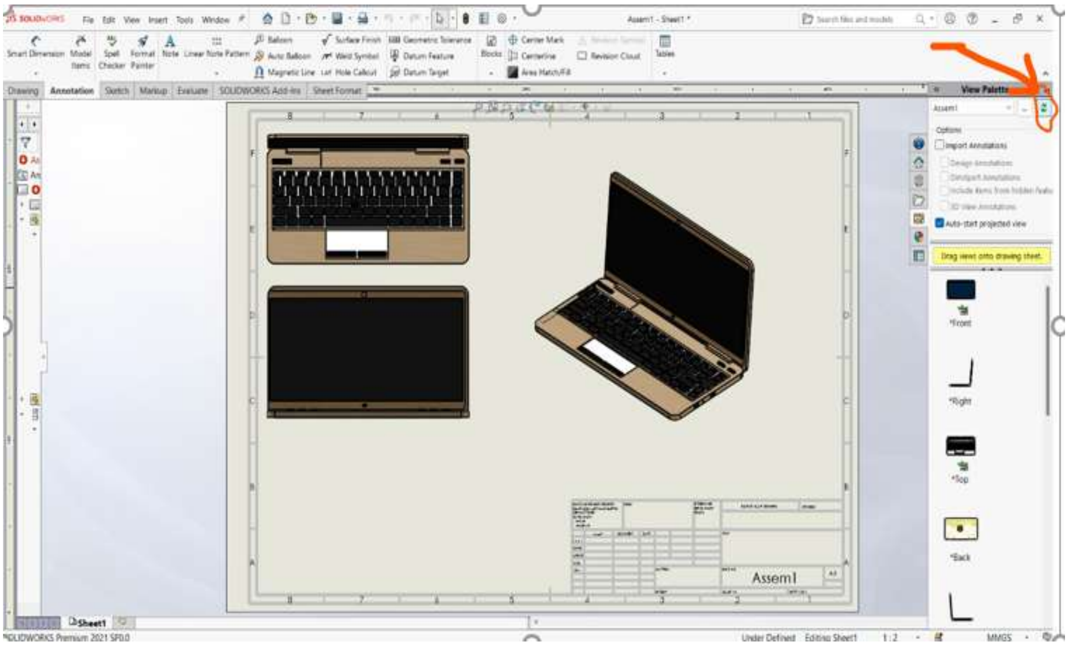
- Choose a part, component or assembly from the file location after clicking on the option shown in the figure below in **red circle**



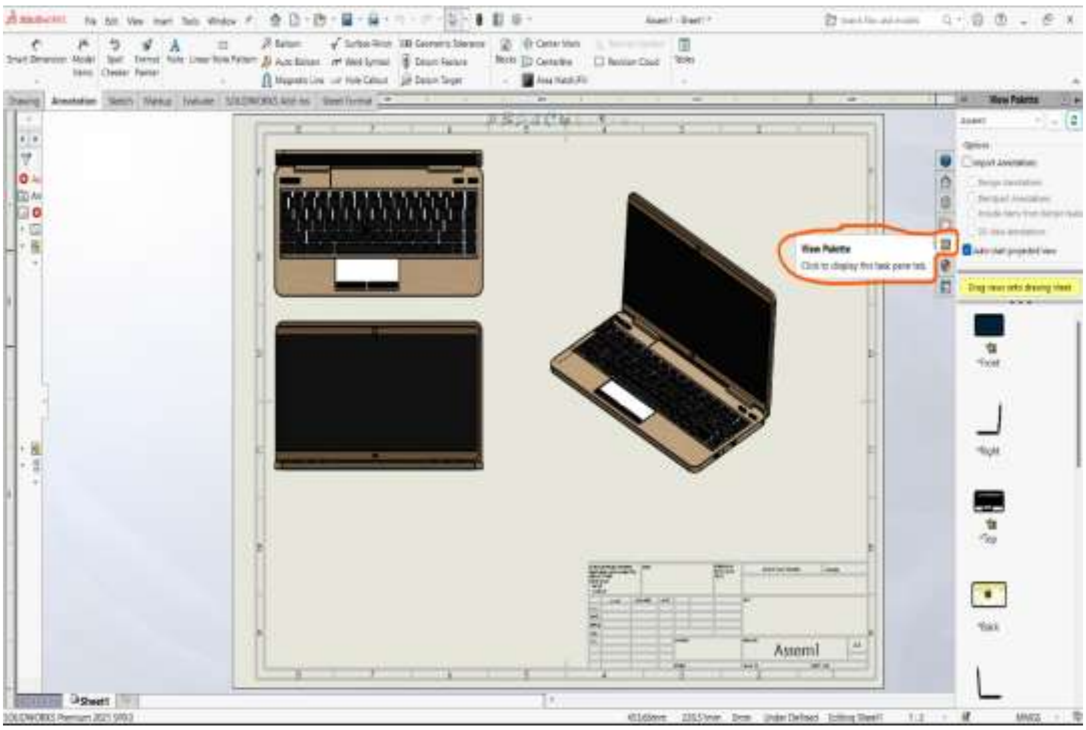
- Position your part, component or assembly views and click **Esc key** on your keyboard



- **Refresh Views**: Update the view palette when changes are made to the model or new views are added.



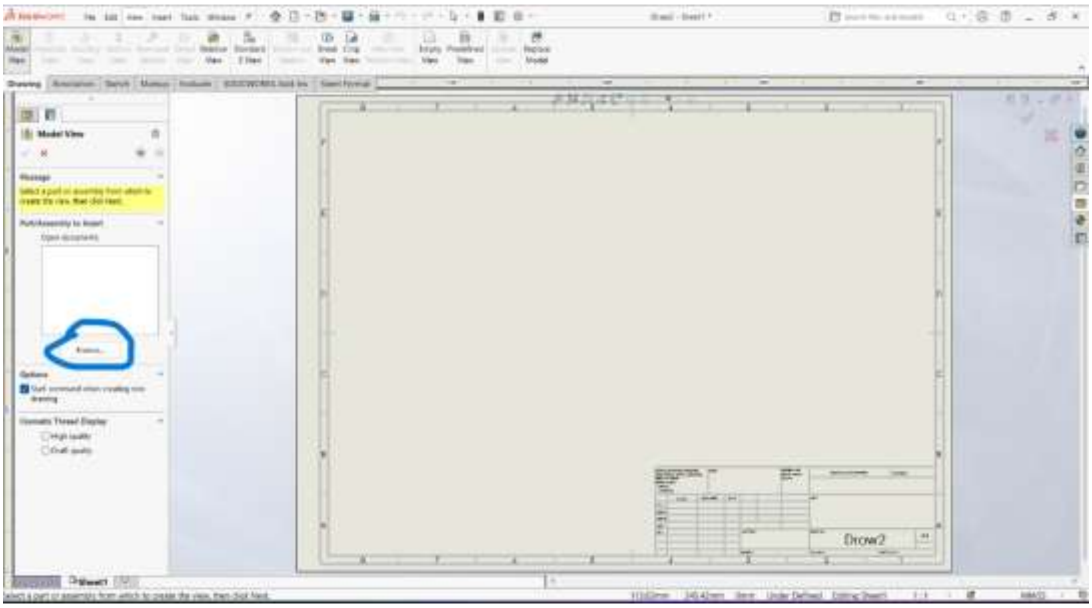
- **Customize Palette**: Add, remove, or rearrange views, and apply filters to narrow down views by type or orientation.
- **Filter views**: You can apply filters to the view palette to narrow down the list of available views based on specific criteria, such as view type, orientation, or custom properties.



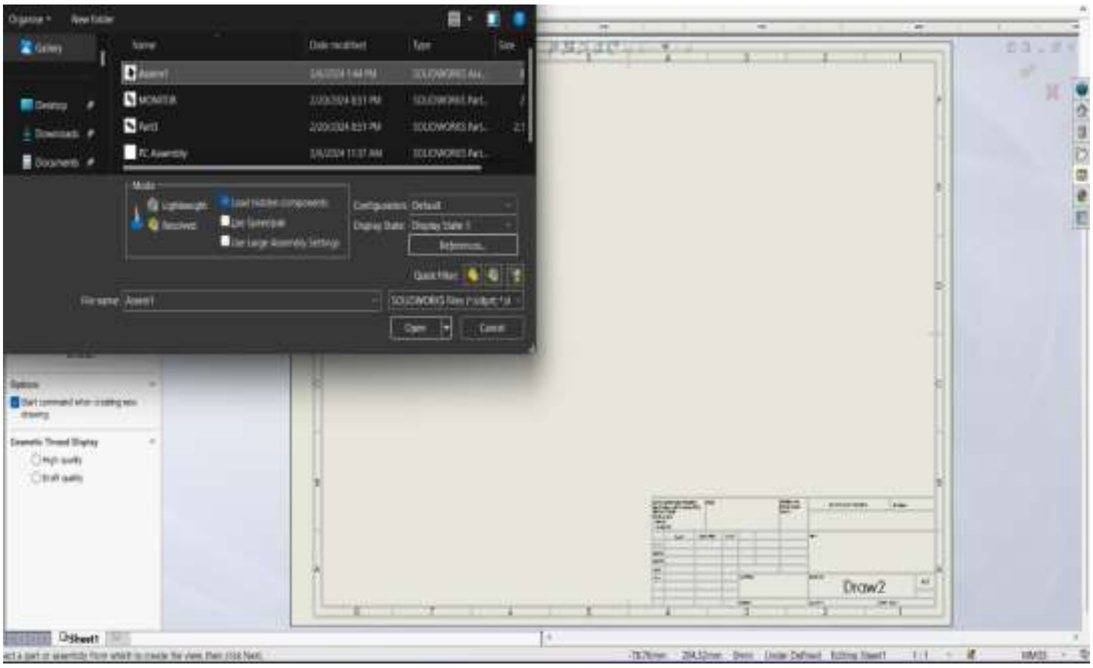
- **Model Views**:

o Open SolidWorks and choose **Drawing template**

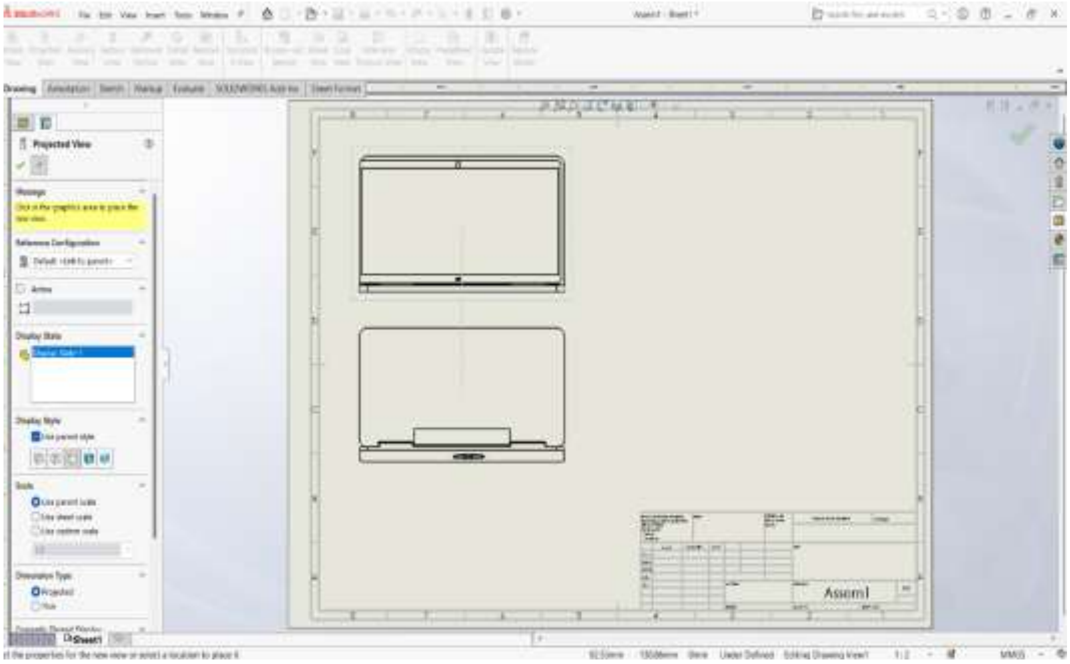
- Click on **Browse**



➢ Choose a part, component or assembly from the file location



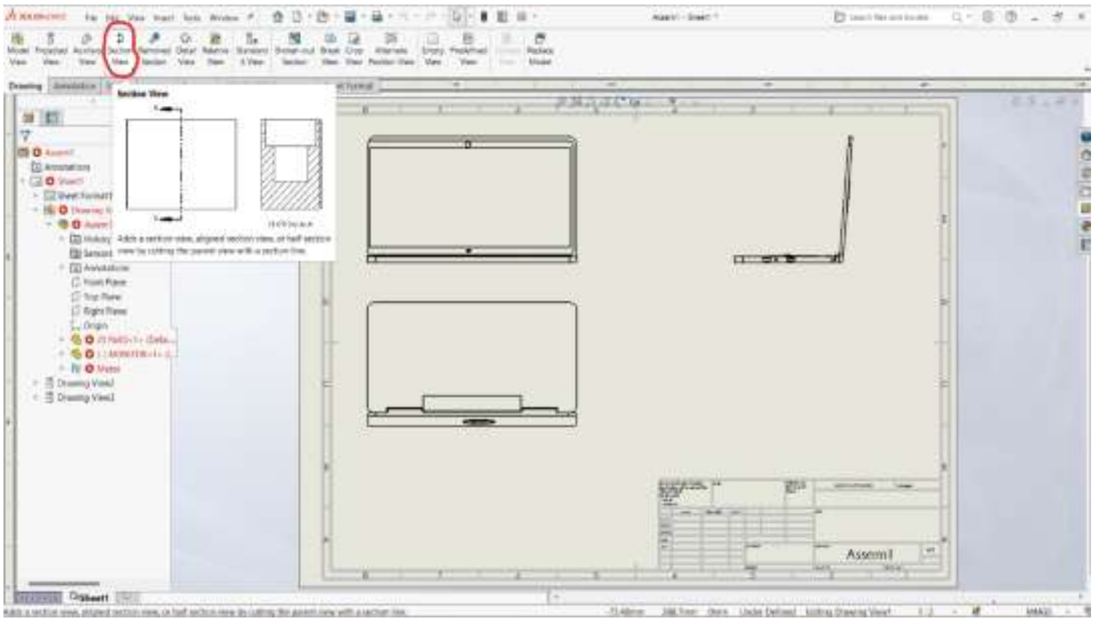
- Insert the views of your part, component or assembly and position them then press **Esc key** on your keyboard



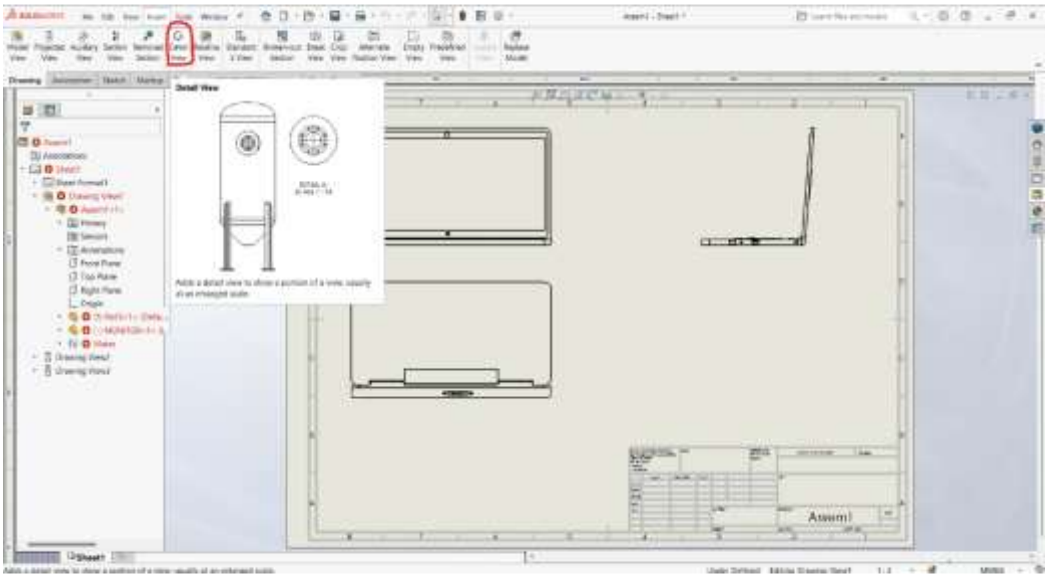
✔ **Standard 3D Views**: Import common views like isometric, front, top, right, and others using shortcut keys (e.g., **Ctrl + 1 for Front View**).

✔ **Custom 3D Views**: Create and import custom views by rotating or zooming in on the model for specific perspectives.

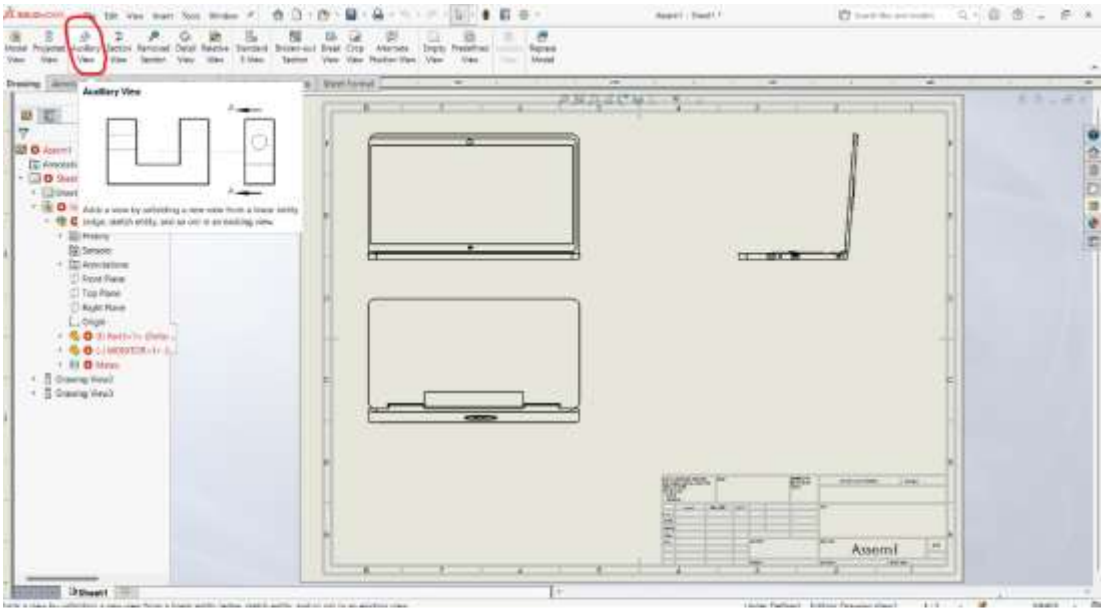
✔ **Section Views**: Import views that cut through the model to show internal details.



✔ **Detail Views**: Magnify a specific area of the model to highlight features.



✔ **Auxiliary Views**: Show the model from alternative angles for additional information.



Points to Remember

- Importing views in SolidWorks allows you to display different perspectives of your model in a drawing.

Application of learning .4.3

QX company needs to create a drawing of a 3D part of a computer case in SolidWorks for manufacturing purposes and you are hired to import multiple views of the 3D part into the drawing sheet. Make sure that you position the views appropriately on the drawing sheet and ensure that the views are correctly aligned, with proper scaling and dimensions.

Indicative content 4.4: Applying Visualization

Duration: 4 hrs

Theoretical Activity 4.4.1: Description of visualization in SolidWorks

Tasks:

1: Refer to the introduction read and answer the following questions:

i. What are the types of drawing views?

ii. What do you understand by the term “Annotation” in SolidWorks?

iii. What are the benefits of using section views and isometric views in SolidWorks when visualizing complex assemblies?

2: Provide the answers for the asked questions

3: Present your findings on the paper/flipchart.

4: Read key reading **4.4.1** for further understanding and ask clarification where necessary

Key readings 4.4.1.: Description of visualization in SolidWorks

In SolidWorks, applying visualization is an essential part of creating detailed and informative drawings.

- **Types of drawing views**: SolidWorks offers various types of drawing views that allow you to represent your model from different perspectives. Some common types of drawing views include:

✔ **Orthographic views**: These views show the model from different directions, such as front, top, right, left, bottom, and back, providing a comprehensive representation of the model.

✔ **Isometric views**: Isometric views display the model in a three-dimensional perspective, showing all three axes equally foreshortened.

✔ **Section views:** Section views are used to reveal internal details of the model by cutting away a portion of the model along a specified cutting plane.

✔ **Detail views:** Detail views are used to magnify a specific area or feature of the model to show intricate details.

✔ **Auxiliary views**: Auxiliary views are created to provide additional information about the model by showing it from a different angle or orientation.

● **Annotation**: Annotation in SolidWorks refers to adding dimensions, notes, symbols, and other graphical elements to the drawing to convey important information. Some common annotation tools and features in SolidWorks include:

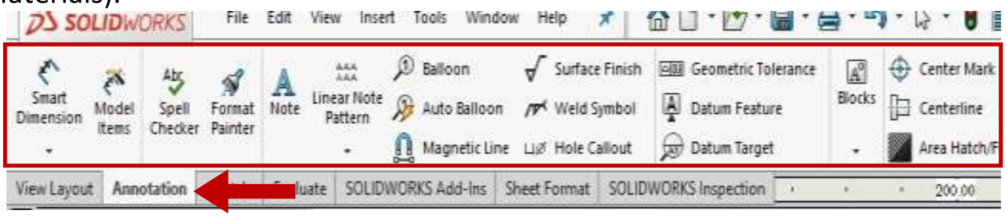
✔ **Dimensions**: Dimensions are used to specify the size and location of features on the drawing. You can add linear dimensions, angular dimensions, radial dimensions, and more.

✔ **Notes**: Notes are textual annotations that provide additional information or instructions related to the drawing. You can add notes using text boxes or callouts.

✔ **Symbols**: Symbols are graphical representations used to convey specific meanings or instructions. SolidWorks provides a library of standard symbols that can be easily added to the drawing.

✔ **Balloons**: Balloons are used to identify and label specific components or parts in an assembly.

They are commonly used in assembly drawings to create parts lists or BOMs (Bill of Materials).



● **Display:** The display options in SolidWorks allow you to control the visibility and appearance of components, features, and annotations in the drawing. Some display options include:

✔ **Hide/show components:** You can selectively hide or show individual components in the drawing to focus on specific areas or details.

✔ **Display states:** Display states allow you to save and switch between different configurations of component visibility, appearance, and display settings.

✔ **Layer control:** Layers can be used to organize and control the visibility of different elements in the drawing, such as dimensions, text, and symbols.

Practical Activity 4.4.2: Applying visualization in SolidWorks

Task:

1: Refer to the introduction read and answer the following questions:

By using a component/part of your choice, after importing views in drawing template you are asked to apply different annotation tools and features that can be used during visualizing your drawing in SolidWorks and also apply different display styles used in SolidWorks when visualizing a drawing?

2: : Read key reading **4.4.2** in trainee manual

3: Pay close attention on trainers demonstration of how to apply visualization.
4: Simulate the steps observed from trainer and apply visualization.
5:Ask for clarification if any

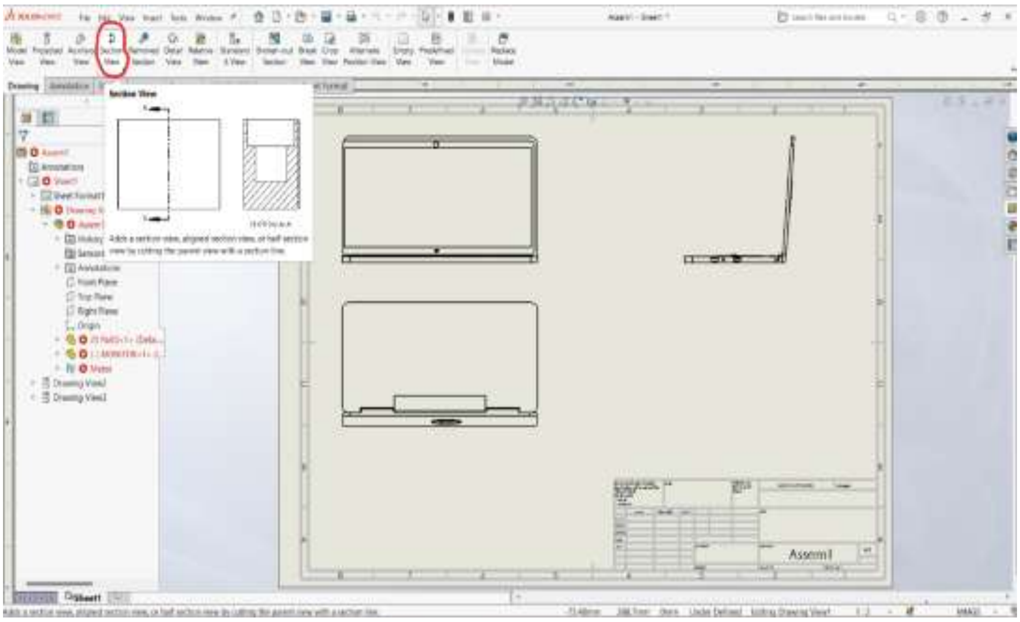
Key readings 4.4.2. Applying visualization in SolidWorks

● **Types of drawing views**: SolidWorks offers various types of drawing views that allow you to represent your model from different perspectives. Some common types of drawing views include:

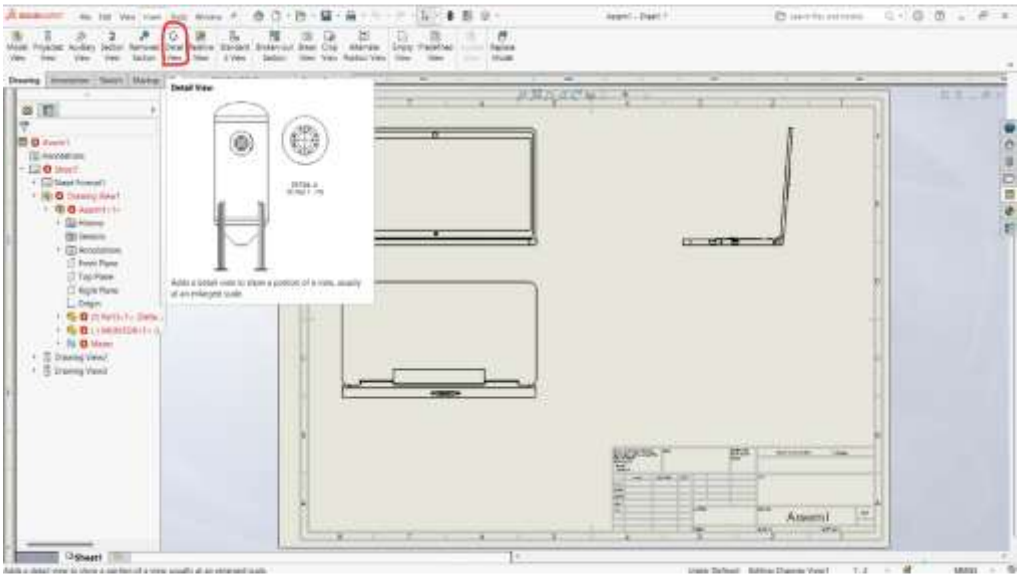
✔ **Orthographic views**: These views show the model from different directions, such as front, top, right, left, bottom, and back, providing a comprehensive representation of the model.

✔ **Isometric views**: Isometric views display the model in a three-dimensional perspective, showing all three axes equally foreshortened.

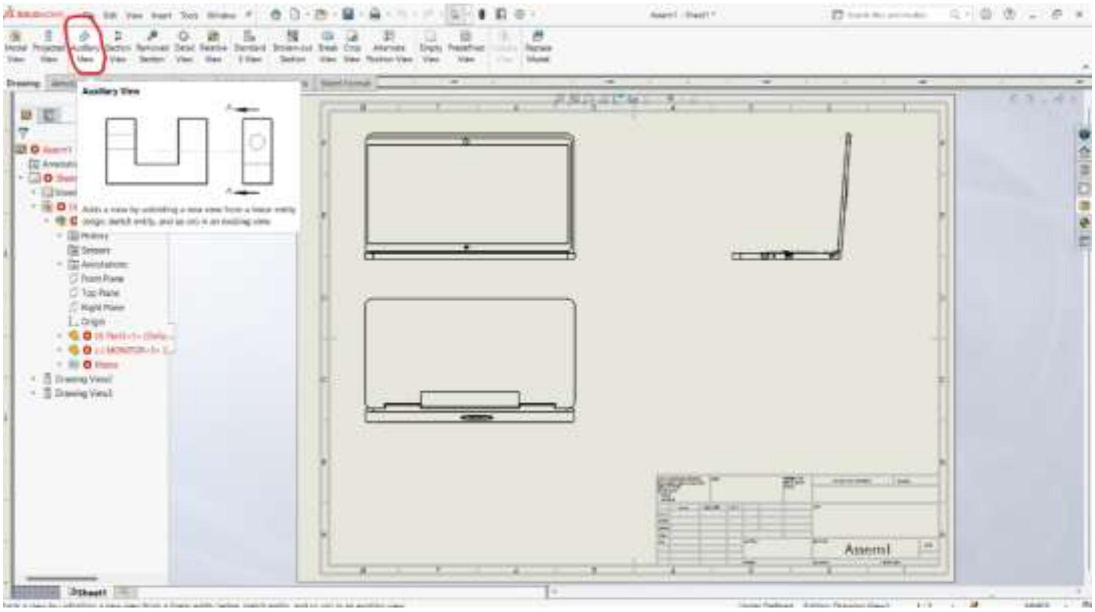
✔ **Section views:** Section views are used to reveal internal details of the model by cutting away a portion of the model along a specified cutting plane.



✔ **Detail views:** Detail views are used to magnify a specific area or feature of the model to show intricate details.



✔ **Auxiliary views**: Auxiliary views are created to provide additional information about the model by showing it from a different angle or orientation.



- **Annotation**: Annotation in SolidWorks refers to adding dimensions, notes, symbols, and other graphical elements to the drawing to convey important information. Some common annotation tools and features in SolidWorks include:

- **Dimensions**

  Steps for applying dimensions:

  - Click **Annotations → Smart Dimension**.
  - Select the edges, faces, or features you want to dimension.
  - Place the dimension by clicking on the drawing sheet.

  You can adjust the dimension properties, like **tolerance**, **precision**, or **units**, by **double-clicking** the dimension and editing the properties in the **PropertyManager**.

- **Notes**

  Steps for applying Notes:

  - Click **Annotations → Note**.
  - Click on the drawing sheet to place the note.
  - Type your text in the pop-up box. You can format the text, change the font, or apply borders from the options in the **PropertyManager**.

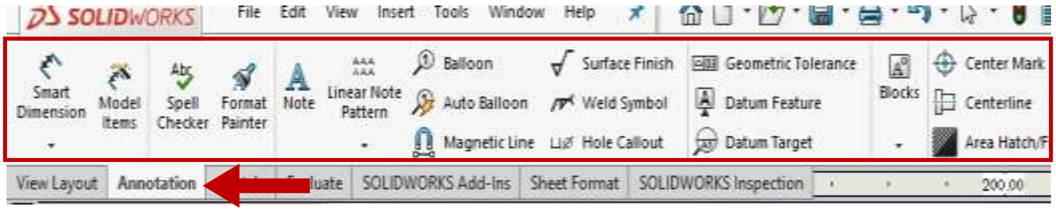
- **Symbols**

  Surface finish symbols specify the quality of the surface finish on a part.

- Go to **Annotations → Surface Finish or Welding symbols** .
- In the **Property Manager** set parameters like size and length details.
- Place the symbol on the relevant part of the drawing

- **Balloons**

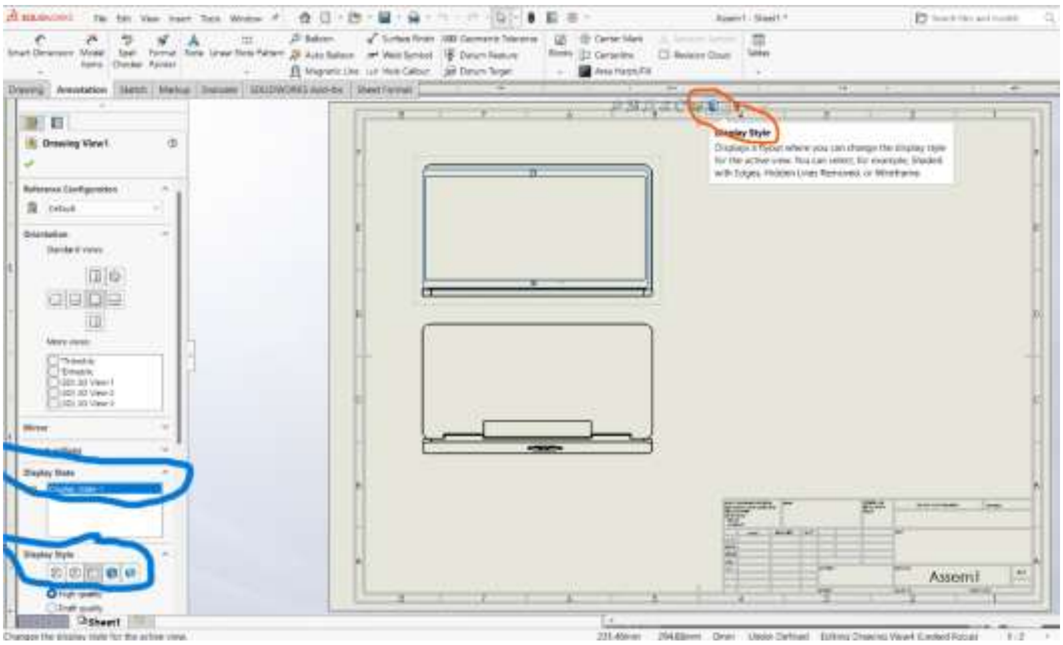
- Click **Annotations → Balloon**.
- Select the part or component in the drawing view.
- The balloon will display the item number, which corresponds to the BOM.



✔ **Display:** The display options in SolidWorks allow you to control the visibility and appearance of components, features, and annotations in the drawing. Some display options include:

• Hide/show components: You can selectively hide or show individual components in the drawing to focus on specific areas or details.

• Display states: Display states allow you to save and switch between different configurations of component visibility, appearance, and display settings.



Points to Remember

- Dimension is most important elements in CAD drawings you must know the elements of diamensions.

Application of learning .4.4

A designer is preparing a model presentation for a client and needs to enhance the model's appearance using various visualization tools in SolidWorks. Your job is to apply visualization techniques to improve the model’s appearance(color, texture, and material properties),add **Section View** for a clearer view of internal components. and aesthetics. In addition you have to provide screenshots of the model with the different visualization techniques applied.

Indicative content 4.5: Exporting the drawing

Duration: 3 hrs

Theoretical Activity 4.5.1: Description of Exporting drawing in SolidWorks

Tasks:

1: **Refer to the introduction read the following questions and provide answers:**

i. What is the purpose exporting drawing to file formats like PDF, DWG, DXF, and JPEG in SolidWorks?

ii. When should you use a PDF file format when exporting a SolidWorks drawing?

iii. Discuss on why it is important to choose the correct export format when sharing drawings with manufacturers or clients?

2: Provide the answer for the asked questions and write them on papers.

3: Present the findings/answers to the whole class.

4: Pay attention to the expert view of trainer.

5: For more clarification, read the key readings 4.5.1 in trainee manual.

Key readings 4.5.1.: Description of Exporting drawing in SolidWorks

Once you have completed your drawing in SolidWorks, you may need to export it for various purposes, such as sharing it with others or printing it. Here are the options for exporting the drawing:

- **File location:** When exporting the drawing, you can choose the desired file location where you want to save the exported file. This can be a local folder on your computer, a network drive, or any other location accessible to you.
- **File format:** SolidWorks offers several file formats to export your drawing, depending on your specific requirements. Some common file formats for exporting drawings include:

✔ SolidWorks Drawing (*.**slddrw**, *.**drw**): This is the native file format for SolidWorks drawings. Exporting your drawing as a SolidWorks Drawing file allows others to open and edit it in SolidWorks.

- ✔ AutoCAD Drawing (*.**dwg**): This is a widely used file format for 2D drawings, compatible with AutoCAD and other CAD software. Exporting your drawing as a DWG file allows others to open and work with it in AutoCAD.

- ✔ Drawing Exchange Format (*.**dxf**): DXF is another commonly used file format for 2D drawings, compatible with various CAD software. It allows for the exchange of drawing data between different programs.

- ✔ Portable Document Format (*.**pdf**): Exporting your drawing as a PDF file ensures that it can be easily viewed and printed on any platform or device without the need for specialized software. PDF files also preserve the formatting and layout of the drawing.

- ✔ Tagged Image File Format (*.**tif**): TIFF files are raster image files that can be exported from SolidWorks. This format is useful when you need a high-quality image of your drawing for documentation or presentation purposes.

- ✔ Joint Photographic Experts Group (*.**jpeg**): JPEG is a commonly used image format that allows for compressed files with good image quality. Exporting your drawing as a JPEG file is suitable when you need to share the drawing as an image.

- ✔ Portable Network Graphics (*.**png**): PNG is another image format that supports lossless compression. It is suitable for exporting drawings with transparency or when you need a high-quality image with a smaller file size compared to TIFF or JPEG.

Practical Activity 4.5.1 Exporting drawings in SolidWorks

Task:

1: Read carefully and answer the following questions

i. How can you export a SolidWorks drawing to DWG or DXF format? What considerations must you keep in mind during this process?

ii. How would you export a SolidWorks drawing to a 3D PDF, and what features does a 3D PDF provide compared to a standard 2D PDF?

2: Provide the answers to the above questions

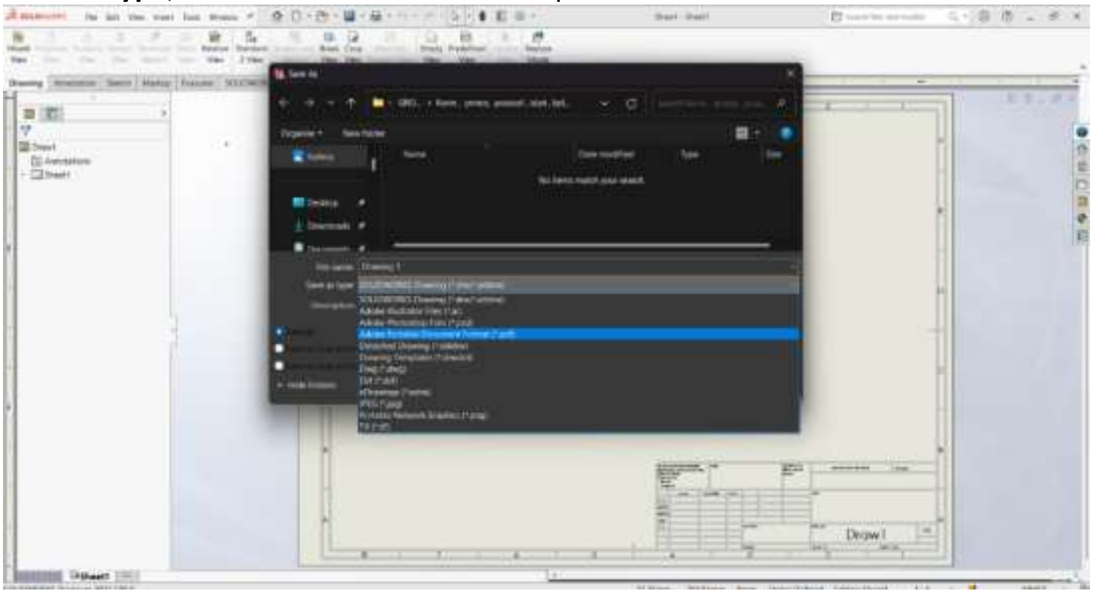
3: Read **Key readings 4.5.2**

4: Insert any component into SolidWorks drawing template and export it into any format that you want

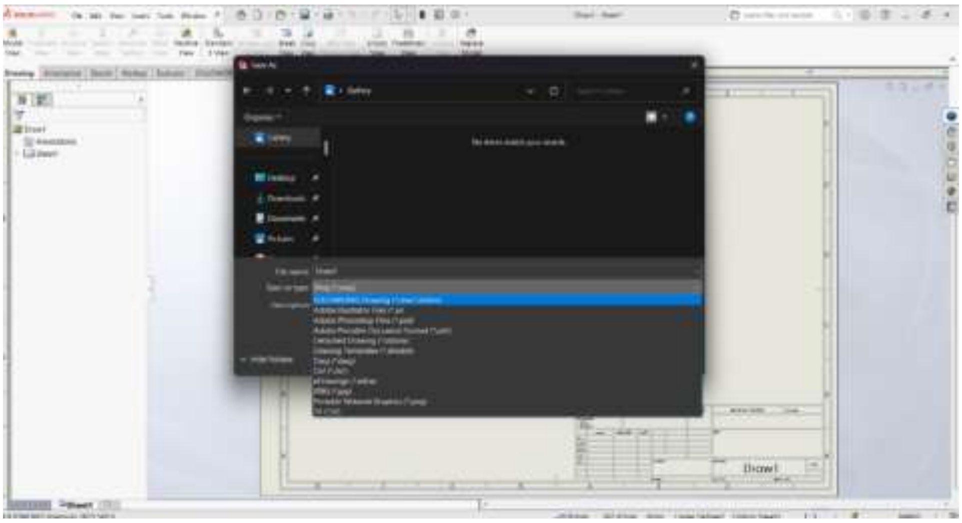
5: In addition, ask questions where necessary.

Key readings 4.5.2 Exporting a Drawing in SolidWorks

- **Method1**
  - **Open the Drawing in SolidWorks:**
  - **Launch SolidWorks**: Open SolidWorks from your desktop or program menu.
  - **Open an Existing Drawing**: To export a drawing, you must first have it open. Click **File** > **Open** and browse to the desired drawing file (.SLDDRW) you want to export.
  - **Prepare the Drawing for Export and Adjust the Sheet Size and Scale (Optional)**
  - **Save** the Drawing with a format of your choice in the **Save As** dialog box, under **Save as type**, select **a format** from the dropdown list.

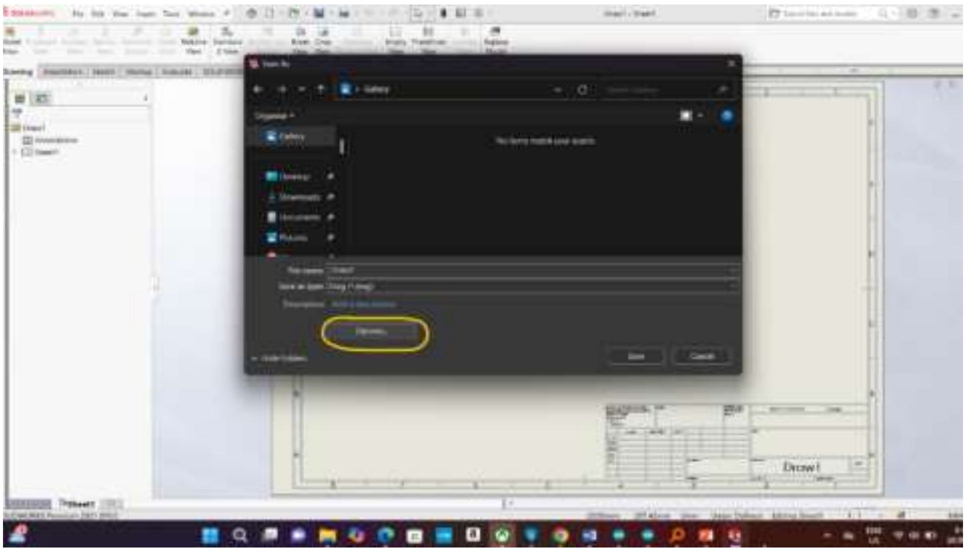


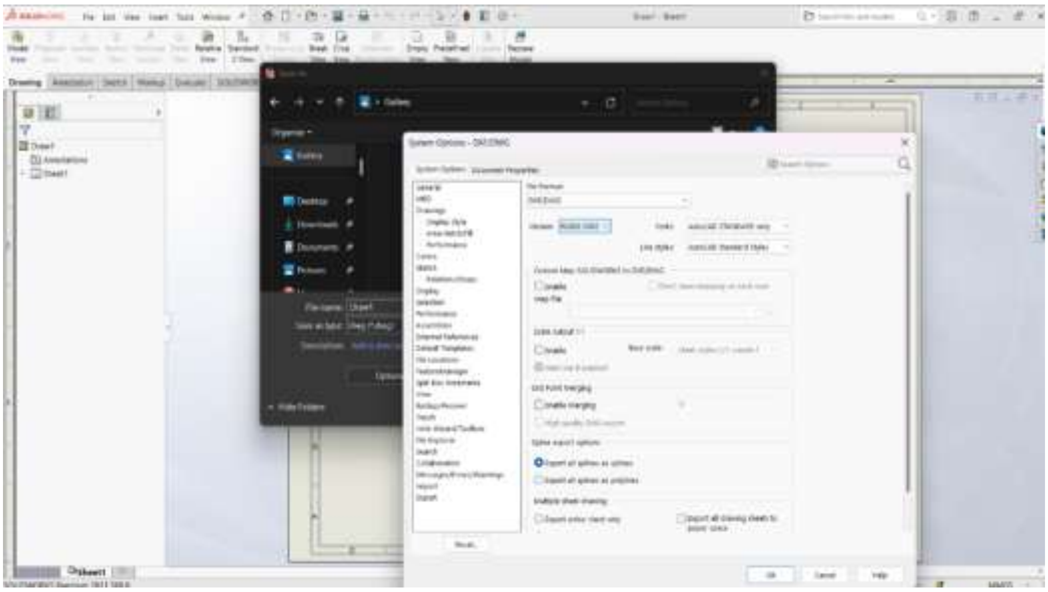
- **Method 2 :Export the Drawing as a DXF or DWG (For CAD Software):**
  - Go to File > Save As: Click File and then select Save As.
  - Choose DXF/DWG as File Type: In the Save As dialog box, select either DXF or DWG as the file type from the dropdown menu.



- Set DXF/DWG Export Options: Click the Options button to configure the export settings:
  Version: Choose the version of AutoCAD for compatibility (e.g., AutoCAD 2013, AutoCAD R12).

  Scale output 1:1: This ensures that the dimensions are exported accurately.





Preview and Confirm the Output: SolidWorks will show a preview window to check the layout of the DXF/DWG file. Ensure everything looks correct before proceeding.

- Save the DXF/DWG: Select the file location, give the file a name, and click Save.
- **Export as an Image File (JPEG, PNG, TIFF, etc.):**
  Go to File > Save As

  Choose the Image Format(From the Save as type dropdown, select the desired image format, such as JPEG, PNG, TIFF, or BMP.

  Set Image Quality Options: Click Options to adjust image settings such as resolution (DPI), size, and color depth.

  Save the Image File: Select the file location, give it a name, and click Save..

Points to Remember

- You must know how to choose where to save the exported file, such as a local folder or network drive.

File Format Options:

- ✔ SolidWorks Drawing (*.slddrw, .drw)*
- ✔ AutoCAD Drawing (*.dwg)
- ✔ Drawing Exchange Format (*.dxf)
- ✔ Portable Document Format (*.pdf)
- ✔ Tagged Image File Format (*.tif
- ✔ Joint Photographic Experts Group (*.jpeg)
- ✔ Portable Network Graphics (*.png)

Application of learning 4.4

You are a technician at a company that designs electronic components. Your team has recently completed detailed technical drawings for a new component, and you are tasked with preparing these drawings for a product design review with both internal team members and external clients.

You need to export the SolidWorks drawings in different formats to meet various requirements for the review process. You will create a PDF for internal review, DXF/DWG files for integration with other CAD systems, and image files for visual presentations.

Learning outcome 4 end assessment

Written assessment

Choose the correct Answer:

1. What do you understand by the term Title Block?

a) A decorative border around the drawing
b) A block containing the main views of the part
c) A section of the drawing that provides essential information about the drawing
d) A tool used to measure dimensions

2. What essential information should be included in a Title Block, and how does it aid in identifying the drawing?

a) Part name, author, date, and drawing number, and it helps identify the drawing by providing clear and concise information
b) Only the part name and date, and it helps to simplify the drawing
c) Scale and color of the part, and it helps control the size of the drawing
d) File location and project notes, helping track revisions

3. How does the customization of the Title Block layout improve communication between the designer and the manufacturer or client?

a) By making the drawing look more professional
b) By adding more details than necessary
c) By allowing specific information relevant to the project to be easily seen and understood
d) By adjusting the size of the drawing sheet

4. Select the correct answer: What is the primary purpose of a Title Block in a SolidWorks drawing?

a) To display 3D views of the part
b) To provide detailed technical specifications
c) To summarize key information about the drawing, such as the part name, author, and scale
d) To control the visibility of different drawing layers

5. Which of the following items is NOT typically included in a standard Title Block?

a) Author’s name
b) Part number
c) Material specifications
d) Drawing scale

6. In SolidWorks, which process can you use to edit the Title Block information in an existing drawing?

a) Right-click on the drawing sheet and select "Edit Sheet"
b) Double-click the Title Block
c) Right-click on the drawing sheet and select "Edit Sheet Format"
d) Select "Insert" from the menu and choose "Title Block"

7. What feature can be used to automatically fill out Title Block fields based on the properties of the part or assembly in SolidWorks?

a) Smart Dimension
b) Custom Properties
c) Bill of Materials (BOM)
d) Annotations

8. What is the purpose of visualization tools in SolidWorks, and how do they contribute to the design process?

a) They add colors to the drawing
b) They help create complex geometries
c) They allow users to preview the design, improving decision-making and presentation
d) They calculate the mass of the parts

9. Explain the difference between Detail view and Auxiliary view in SolidWorks. When would you use each tool?

a) Detail view shows zoomed-in sections of the model, while Auxiliary view provides projections from a slanted plane.
b) Both are used to measure distances in the model.
c) Auxiliary view is only used for circular parts.
d) Detail view and Auxiliary view are not related to the drawing process.

10. How does applying materials and textures to a 3D model in SolidWorks help in visualizing the final product?

a) It creates a photorealistic view of the final product, aiding in communication with clients and testing material properties.

b) It makes the part heavier.
c) It helps avoid making mistakes during assembly.
d) It doesn't impact the design process.

11. What are the benefits of using section views and isometric views in SolidWorks when visualizing complex assemblies?

a) Section views show internal features, while isometric views provide a 3D perspective for better overall visualization.
b) Section views are used for coloring, and isometric views are for cutting.
c) They help add text annotations.
d) They adjust the scaling of the model automatically.

12. A3 paper size is larger than A2 paper size.

a) True
b) False

13. What is the default sheet orientation for most drawings?

a) Landscape
b) Portrait
c) Horizontal
d) Vertical

14. What are the two main orientations of drawing sheets, and when would each be used?

a) Landscape and Portrait; Landscape is used for wide designs, while Portrait is used for taller designs
b) Vertical and Horizontal; Horizontal for large parts and vertical for smaller ones
c) North and South orientations for parts
d) Left and Right orientations for better visibility

15. Explain why different drawing sheet sizes (A0, A1, A3, etc.) are used in drawings?

a) To accommodate different scales and levels of detail required in the project
b) To organize the parts into different sections
c) To avoid confusion when printing
d) To reduce the number of dimensions on a drawing

Practical assessment

As a technician you are tasked with assisting a company in designing a PCB enclosure for a key client using SolidWorks. Your responsibilities include setting up an organized drawing sheet layout, customizing the Title Block with project and client details, and adhering to company standards. You will import multiple views of the 3D part into the drawing sheet, ensuring proper alignment, scaling, and dimensions for manufacturing. Additionally, you will enhance the model's appearance using visualization techniques, including colors, textures, and section views. Finally, you will export the drawings in various formats, such as PDF, DXF/DWG, and image files, for internal and external reviews.

References

GoEngineer. (2022, September 20). *Apply a new SOLIDWORKS drawing sheet format for future drawings*. GoEngineer. https://www.goengineer.com/blog/apply-new-solidworks-drawing-sheet-format-future-drawings

Ji, P. (2011). *SolidWorks essentials*. Concord, Massachusetts: Ji Pengcheng https://www.scribd.com/doc/194529688/SolidWorks-Essentials-Ver-2011

RTB | RWANDA TVET BOARD

October, 2024