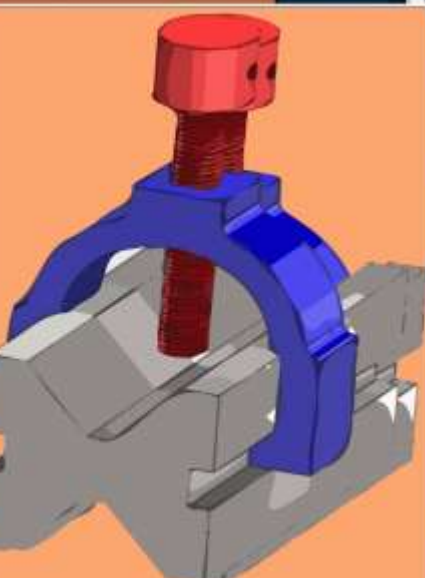
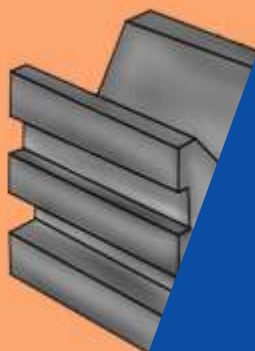




RQF LEVEL 4



GENSD401
MANUFACTURING
TECHNOLOGY

SolidWorks
Design

TRAINER'S MANUAL

October, 2024



SOLIDWORKS DESIGN



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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 Manufacturing Technology, specifically for the module **"GENSD401: SolidWorks Design"**

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



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ACRONYMS

16 GB: Six-teen Gigabytes

2D: Two Dimension

3D : Three Dimension

3D PDF : 3D Portable Document Format

ACIS: Andy, Charles, Ian's System

CAD: Computer-Aided Design

CADD: Computer-Aided Design and Drafting

CAE: Computer-Aided Engineering

CAM: Computer-Aided Manufacturing

CBT/A: Competency-Based Training and Assessment

CD drive: Compact Disc

CPU: Central Processing Unit

DVD: Digital Video Disc

GUI: Graphical User Interface

IGES: Initial Graphics Exchange Specification

RAM: Random Access Memory

RTB: Rwanda TVET Board

STEP: Standard for the Exchange of Product model data

STL: Stereo Lithography

TQUM Project: TVET Quality Management Project

USB: Universal Serial Bus

INTRODUCTION

This trainer's manual includes all the methodologies required to effectively deliver the module titled "**SolidWorks Design**." 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 trainer'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 trainer, you will begin by asking questions related to the activities to encourage critical thinking and guide trainees toward real-world applications in the labor market. The manual also outlines essential information such as learning hours, didactic materials, and suggested methodologies.

This manual outlines the procedures and methodologies for guiding trainees through various activities as detailed in their respective trainee manuals. The activities included in this training manual are designed to offer trainees opportunities for both individual and group work. Upon completing all activities, you will assist trainees in conducting a formative assessment known as the end learning outcome assessment. Ensure that trainees review the key reading and the points to remember section.

MODULE CODE AND TITLE: GENSD401 SOLIDWORKS DESIGN

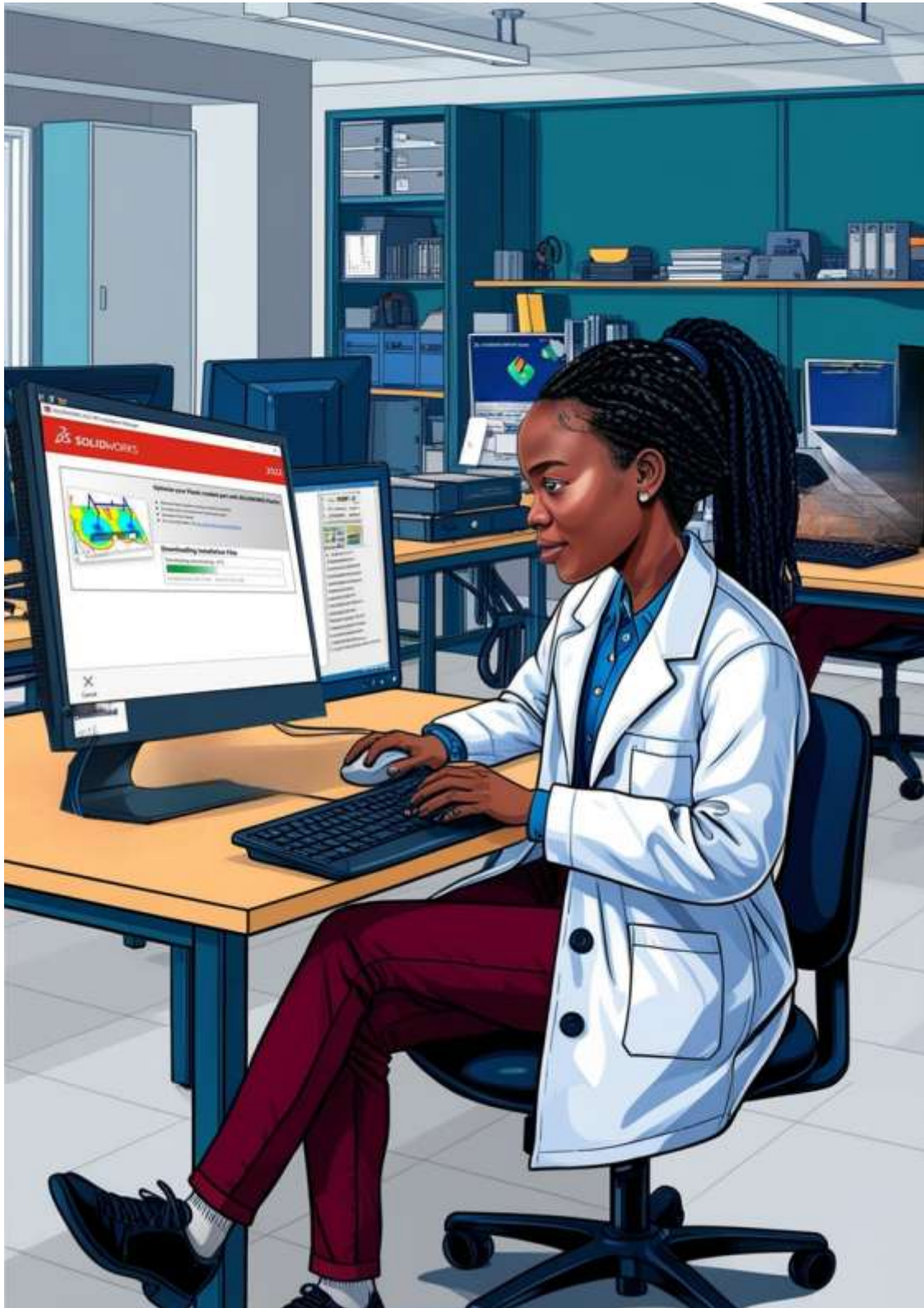
Learning Outcome 1: Set up SolidWorks Space

Learning Outcome 2: Create Parts

Learning Outcome 3: Assemble Parts

Learning Outcome 4: Produce Drawing

Learning Outcome 1: Set Up SolidWorks Space



Indicative contents
1.1 Introduction to SolidWorks software
1.2 Installing SolidWorks software
1.3 Selecting SolidWorks template
1.4 Customizing Interface

Key Competencies for Learning Outcome 1: Set up SolidWorks Space

Knowledge	Skills	Attitudes
• Definition of key terms used in SolidWorks software • Description of the applications of SolidWorks • Description of advantages and disadvantages of SolidWorks software • Identification of SolidWorks software installation steps • Identification of SolidWorks templates • Identification of SolidWorks software interface	• Installing SolidWorks software • Selecting SolidWorks template • Customizing SolidWorks interface	• Being effective time management and organizational skills in SolidWorks installation • Having self confidence in SolidWorks installation • Having a continuous learning and skill development in the Selection SolidWorks template • Having adaptation to software updates and industry requirements • Being aware of collaborative mindset for effective coordination for industry trends • Being proactive to problem-solving and troubleshooting abilities during installation • Having ethical and professional in the

		Customization of SolidWorks interface
 Duration: 10 hrs		
Learning outcome 1 Objectives:  By the end of the learning outcome, the trainees will be able to: 1. Explain clearly the key concepts of SolidWorks as used in SolidWorks software. 2. Install correctly SolidWorks software in a computer used in SolidWorks design. 3. Identify properly SolidWorks templates used in the SolidWorks software 4. Select correctly SolidWorks template used in SolidWorks design 5. Identify properly interface used in SolidWorks software 6. Customize correctly the SolidWorks Interface used in SolidWorks design.		
 Resources		
Equipment	Tools	Materials
• Computers • Projector	• SolidWorks software, • SolidWorks tutorials, • Online Courses, • Flash disk drive, • Hard disc drive, • Internet cable, • internet connections, • CD drive • Mouse	• Student workbook, • Papers, • Textbooks • SolidWorks Documentation, • Exercise Files • Sample Models, • Whiteboard • Visual Aids
 Advance Preparation: Before delivering this learning outcome, you are recommended to: • Avail a computer(s) with sufficient capacity • Prepare computer lab ready to be used		

- Avail the updated SolidWorks software
- Avail good and working projector
- Check the availability of electricity
- Avail tutorial videos to be used as didactic material related to the activity to be done in SolidWorks software



Indicative content 1.1: Introduction to SolidWorks Software



Duration: 2 hrs



Theoretical Activity 1.1.1: Explanation of key concepts of SolidWorks software



Notes to the trainer:

- While delivering this activity, trainer may use small groups for explaining key concepts of SolidWorks software
- For delivering this activity effectively trainer may use the tutorials video related to SolidWorks Software.



Key steps:

While delivering this activity, pass through the following steps:

Step 1: Introduce the session and request trainees to answer the following questions:

- What do you understand by the following terms used in SolidWorks Software?
 - SolidWorks
 - CAD
 - CAM
 - CADD
 - CAE
 - 2D Modeling
 - 3D Modeling
 - SolidWorks Setup
- What should be the applications of SolidWorks software?
- What could be the advantages and disadvantages of SolidWorks software?

Step 2: Ask trainees to write provided answers on flipchart/paper, Blackboard or white board.

Step 3: Engage trainees in presentation of their findings.

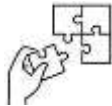
Step 4: Provide expert view and clarify concepts using didactic materials, then address any questions or concerns.

Step 5: Ask trainees to read the key reading 1.1.1. In trainee manual.



Points to Remember

- SolidWorks allows for realistic simulations of material behavior, enhancing design accuracy and quality, and supports easy modifications to 3D models, boosting productivity and collaboration among design teams
- SolidWorks software requires a high-performance computer and comes at a significant cost, which can be a barrier for some users



Application of learning 1.1.

Visit the SolidWorks lab, where you will meet with the lab manager to discuss the uses, advantages, and disadvantages of SolidWorks software in the industrial sector. You are required to complete the following tasks:

- ✓ Prepare a summary on the applications of SolidWorks software.
- ✓ Prepare a summary to the advantages and disadvantages of SolidWorks software

Checklist

SN	Criteria	Indicators	Yes	No
1.	SolidWorks applications are properly explained	1.1. SolidWorks applications are outlined		
		1.2. SolidWorks applications are explained		
2.	Advantages and disadvantages of SolidWorks software are properly explained	2.1. Advantages of SolidWorks are outlined		
		2.2. Advantages of SolidWorks are explained		
		2.3. Disadvantages of SolidWorks are outlined		
		2.4. Disadvantages of SolidWorks are explained		



Indicative content 1.2: Installing SolidWorks Software



Duration: 3 hrs



Practical Activity 1.2.1: Installing SolidWorks software



Notes to the trainer

- Trainer may demonstrate to trainees how to install SolidWorks software. For effective delivery; it is recommended to:
 - ✓ Avail an updated version of the SolidWorks setup
 - ✓ Avail computers with good performance.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to go to the SolidWorks computer lab and install SolidWorks software on computers.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated)
- Step 3:** Demonstrate how to install SolidWorks software properly. While explaining the installation procedures
- Step 4:** Ask trainees to install SolidWorks software by respecting installation procedures.
- Step 5:** Check whether installation procedures are applied and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 1.2.1 in the trainee manual.
- Step 7:** Ask trainees to perform the application of learning 1.2.

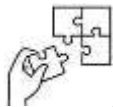


Points to Remember

- Steps for installing SolidWorks software must be followed carefully to avoid installation failure. (refer to installation manual).
- When installing SolidWorks software, you need to choose a computer that is compatible

with the SolidWorks software version you are going to install.

- Disabling antivirus should be temporary. Always re-enable it after installing SolidWorks software.
- If you must disable antivirus, make sure you are not downloading or running files from untrusted sources during this time.
- Do not let your computer shut down while installing SolidWorks because this can disturb installation success.



Application of learning 1.2.

The school has set up a new SolidWorks computer lab equipped with 20 new computers. Ask your trainees to install SolidWorks software on all 20 computers to ensure the lab is ready to begin designing using SolidWorks.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Compatibility of computer is properly checked	1.1. Computer is started		
		1.2 The selected computer is running a compatible version of the operating system		
		1.3. Anti-virus and firewall are temporary disabled		
2.	SolidWorks Software is properly installed	2.1 The Installation Media is obtained.		
		2.2 Installation manager launched		
		2.3 Correct installation type selected		
		2.4 Installation location chosen		
		2.5 Serial number entered		
		2.6 Required add-ins selected		

		2.7 License agreement accepted		
		2.8 Installation started and Finished		
		2.9 SolidWorks software is Launched.		



Indicative content 1.3: Selecting SolidWorks Template



Duration: 2 Hours



Practical Activity 1.3.1: Selecting SolidWorks template



Notes to the trainer

- To support this activity, the trainer can demonstrate to each trainee how to select a SolidWorks template in the SolidWorks software for better understanding; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

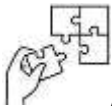
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to go to the computer lab, open the SolidWorks software, and carefully select the appropriate SolidWorks template to design a bolt as required.
- Step 2:** Explain the task and provide clear work instruction (Task, Time allocated)
- Step 3:** Demonstrate how to select SolidWorks template in SolidWorks software properly.
- Step 4:** Ask trainees to select SolidWorks template and monitor the procedures.
- Step 5:** Check whether selection of template is done properly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 1.3.1. in trainee manual.
- Step 7:** Ask trainees to perform the task provided in application of learning 1.3.



Points to Remember

- In SolidWorks software there are three templates.
- Select part template to design a single part or component
- Select Assembly template to make assembling of designed parts or components
- Select drawing template to produce drawing of the designed component or parts.



Application of learning 1.3.

The school plans to design a bolt using SolidWorks software and needs to select the most suitable template. Ask your trainees to choose the appropriate template to ensure the successful design of the bolt.

Checklist:

SN	Criteria	Indicators	Yes	No
1.	Computer is appropriately Prepared	1.1. Computer is started		
		1.2. SolidWorks software is launched		
2.	SolidWorks template is well selected according to the work to be done	2.1. New document opened		
		2.2. SolidWorks template are identified		
		2.3. The Part template is selected and open		



Indicative content 1.4: Customizing Interface



Duration: 3 Hours



Practical Activity 1.4.1: Customising interface



Notes to the trainer

- Trainer may deliver this content by demonstrating trainees how to customize the SolidWorks interface within SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Didactic materials may be used (video tutorials)



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to go in computer Lab, open SolidWorks software and customize SolidWorks interface carefully and successfully as required to facilitate the design of bolt.
- Step 2:** Explain the task and provide clear work instruction (Task, Time allocated)
- Step 3:** Demonstrate how to customize SolidWorks interface properly.
- Step 4:** Asks trainees to customize SolidWorks interface and monitor the procedures.
- Step 5:** Checks whether customization is done properly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 1.4.1. in trainee manual.
- Step 7:** Ask trainees to perform the task provided in application of learning 1.4.



Points to Remember

- The SolidWorks Customization interface allow user to interact with various tools, features, and commands to create, modify, and manage 3D models, assemblies, and drawings.
- It is essential to every step in customizing SolidWorks interface for well achieving customization of an interface.



Application of learning 1.4.

The school want to design a hexagonal headed bolt using SolidWorks software and needs to customise interface. Ask your trainees to customise interface to ensure the successful design of the bolt.

Checklist:

SN	Criteria	Indicators	Yes	No
1.	SolidWorks software are properly Launched	1.1 New document opened		
		1.2. SolidWorks template are identified		
		1.3. The Part template is selected and open		
2.	Interface is properly customized according to the work to be done	2.1. SolidWorks interface are identified		
		2.2. SolidWorks interfaces are customized.		



Learning outcome 1 end assessment

Written assessment

1. Read carefully the following statement and answer by True if the statement is correct or false if the statement is wrong.

i. The following are the explanation of key terms found in SolidWorks software.

- a) **CAD:** Computer-Aided Manufacturing
- b) **CAM:** Computer-Aided Design and Drafting
- c) **CADD:** Computer-Aided Design
- d) **CAE:** ~~Computer~~-Aided Engineering

ANS:

- a) **FALSE**
- b) **FALSE**
- c) **FALSE**
- d) **TRUE**

ii. SolidWorks is used to design mechanical components and systems, including machinery, automotive parts, and aerospace components.

ANS: TRUE

iii. The software cannot simulate manufacturing processes such as injection molding, machining, and welding.

ANS: FALSE

iv. SolidWorks helps users shorten their design process by eliminating repetitive activities.

ANS: TRUE

v. SolidWorks does not require a computer system with sufficient hardware resources to run smoothly.

ANS: FALSE

vi. When installing SolidWorks software, we do not need to disable Antivirus/Firewall because some antivirus programs or firewalls may interfere with the installation process.

ANS: FALSE

vii. In SolidWorks toolbar provides quick access to frequently used commands and tools.

ANS: TRUE

viii. In SolidWorks, the workspace refers to the area where you store your design and modelling tasks.

ANS: FALSE

ix. Keyboard Shortcuts Is a combination of keys that, when pressed together, perform a specific command or action.

ANS: TRUE

2. Fill the empty space of the following sentences to make them meaningful by using the following terms: **Mechanical Engineering, Accept the License Agreement, Part Templates, Assembly Templates, and Drawing Templates.**
- i. SolidWorks is applied in various industries such as **Mechanical Engineering...**
 - ii. **License Agreement.....** Outlines the terms and conditions of using the software.
 - iii. ...**Part Templates.....** Are used when creating individual 3D models of a specific part or component
 - iv. **Assembly Templates.....** Are used when creating assemblies, which are collections of multiple parts and sub-assemblies that fit together to form a larger product.
 - v.**Drawing Templates.....** Are used when creating 2D drawings of the parts and assemblies that you have modeled in SolidWorks
3. Choose the letter corresponding to the right answer by referring to the term **bolded** with its key shortcut.
- I. In SolidWorks software, the short cut which serve as “**create New**” is:
 - a) Ctl+V
 - b) Ctl+Y
 - c) Ctl+N**ANS: C**
 - II. In SolidWorks software, the short cut which serve as “**open**” is:
 - a) Ctl+R
 - b) Ctl+S
 - c) Ctl+O**ANS: C**
 - III. In SolidWorks software, the short cut which serve as “**Print**” is:
 - a) Ctl+M
 - b) Ctl+W
 - c) Ctl+P**ANS: C**
 - IV. In SolidWorks software, the short cut which serve as “**Undo**” is:
 - a) Ctl+C
 - b) Ctl+A
 - c) Ctl+Z**ANS: C**

Practical assessment

The school aims to design ISO hexagonal-headed bolts using SolidWorks software. Ask your trainees to perform the following tasks:

- ✓ Install SolidWorks software on your computer.
- ✓ Choose the appropriate SolidWorks template.
- ✓ Customize the interface.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Solid Works software is properly installed according to the manufacturer's instructions.	1.1. SolidWorks software setup is Installed		
		1.2 SolidWorks software is Launched.		
2.	Template is correctly selected according to the work required.	2.1 New document opened		
		2.2. The Part template is selected and open		
3.	Interface is properly customized according to the work to be done	3.1. SolidWorks interface are identified		
		3.2. MGS(units) selected		
		3.3. SolidWorks interfaces are customized.		



Further information to the trainer

Cadartifex. (2021). A Power Guide for Beginners and Intermediate Users. SolidWorks.

Musto, W. H. (n.d.). Introduction to Solid Modeling Using SolidWorks.

Planchard, D. P. (n.d.). Engineering Design with SolidWorks.

Weber, G. V. (2021). Black Book. SolidWorks.

Learning Outcome 2: Create Parts



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 Parts

Knowledge	Skills	Attitudes
• Description of plane used in SolidWorks software • Description of feature tools used in SolidWorks • Identification of SolidWorks tools • Description the geometrical sketch • Description of application of various visualisation in SolidWorks	• Selecting of SolidWorks plane and layout • Creating geometrical sketch for SolidWorks software • Taking measurement of objects • Applying dimensions and text • Applying part features of SolidWorks software • Applying visualization of SolidWorks software • Exporting part used for filing SolidWorks projects • Performing SolidWorks basic settings	• Having creativity and innovation throughout the design works • Having attention to design details • Being resourcefulness in the new design trends • Being persistent, team working, flexibility, punctuality during project design



Duration: 30 hrs

Learning outcome 2 objectives:



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

1. Describe properly SolidWorks planes used in SolidWorks design.
2. Select correctly SolidWorks plane used in SolidWorks design.
3. Perform correctly basic settings used in SolidWorks software
4. Apply properly dimensions of geometrical objects used in SolidWorks software
5. Create correctly geometrical sketches used in SolidWorks software
6. Describe properly feature tools used in SolidWorks software
7. Apply correctly SolidWorks visualization used in SolidWorks design.
8. Export correctly designed part as used in SolidWorks design.



Resources

Equipment	Tools	Materials
• Computer • Projector	• SolidWorks software, • SolidWorks tutorials, • Online Courses, • Flash disk drive, • Hard disc drive, • Internet cable, • internet connections, • CD drive • Mouse	• Student workbook, • Papers, • Textbooks • SolidWorks Documentation, • Exercise Files • Sample Models, • Whiteboard • Visual Aids



Advance Preparation:

Before delivering this learning outcome, you are recommended to:

- Avail installed computer for SolidWorks
- Avail hard sketches of tutorials.
- Prepare computer lab ready to be used
- Have an updated SolidWorks software
- Have a good and working projector
- Check the availability of electricity
- Avail tutorial videos to be used as didactic material related to the activity to be done in SolidWorks software.



Indicative content 2.1: Selecting of Plane



Duration: 3 hrs



Theoretical Activity 2.1.1: Description of plane used in SolidWorks software



Notes to the trainer:

- While delivering this activity, small group can be used for describing planes used in SolidWorks software
- The use of images, videos, and illustrations as didactic materials is required.



Key steps:

While delivering this activity, pass through the following steps:

Step 1: Introduce the activity and request trainees to describe the following planes used in SolidWorks software:

- i. Standard plane
- ii. Custom plane

Step 2: Ask trainees to provide and write their answers on flipchart/paper.

Step 3: Engage trainees in presentation of their findings.

Step 4: Provide expert view using didactic materials and address questions or Concerns.

Step 5: Ask trainees to read the key reading 2.1.1. In trainee manual.



Points to Remember

- In the SolidWorks software, there is Two types of planes, Standard and Custom Planes.
- Selecting a plane in SolidWorks involves choosing a reference plane (such as the Front, Top, or Right Plane) or creating a custom plane to start your sketch or feature. When deciding whether to use a standard plane or create a custom plane in SolidWorks, the choice depends on your design requirements and the specific features you need to create.



Practical Activity 2.1.2: Selecting of plane used in SolidWorks software



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee on how to select the plane in SolidWorks software to aid in the design of hexagonal headed bolt. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

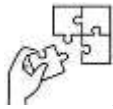
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to open SolidWorks software select the appropriate planes in SolidWorks to aid in the design of bolt.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to select the appropriate planes in SolidWorks design based on the given project.
- Step 4:** Ask trainees to select the appropriate planes during SolidWorks design and monitor the procedures.
- Step 5:** Check whether selection of plane is done correctly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 2.1.2.
- Step 7:** Ask trainees to perform the application of learning 2.1.



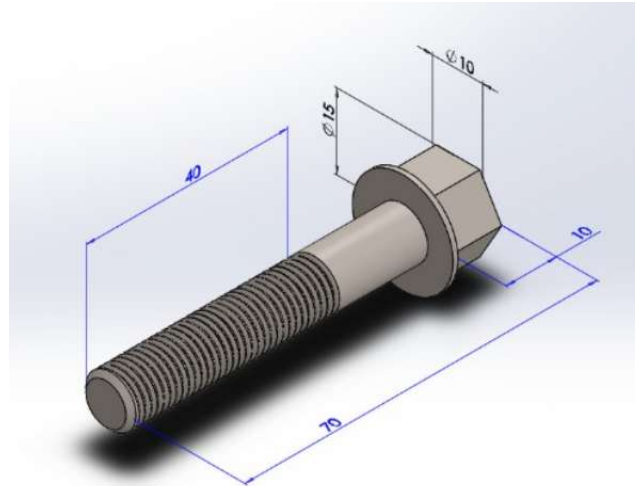
Points to Remember

- When deciding whether to use a standard plane or create a custom plane in SolidWorks, the choice depends on your design requirements and the specific features you need to create.
- Selecting the wrong plane in SolidWorks can lead to a misaligned design, making it difficult to accurately place and orient sketches or features. This can cause errors in the assembly process, leading to conflicts and incorrect part relationships



Application of learning 2.1.

Ask your trainees to identify the appropriate plane where the sketch for the hexagonal-headed bolt should be initiated, based on the provided drawing.



Checklist

SN	Criteria	Indicators	Yes	No
1.	SolidWorks software is appropriately Runned	1.1. SolidWorks is lunched		
		1.2. Appropriate template is selected according to the work to be done.		
2.	SolidWork plane is correctly selected according to the work.	2.1. Planes are identified		
		2.2. Plane is selected		



Indicative content 2.2: Creating Sketch



Duration: 10 hrs



Practical Activity 2.2.1: Creating sketch in SolidWorks software



Notes to the trainer

- Facilitation of this activity, trainer can demonstrate individual trainee how to create sketch in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

While delivering this activity, pass through the following steps:

Step 1: Introduce the activity and ask trainees to create sketch which will be used to design a bolt by SolidWorks software.

Step 2: Explain the task and provide clear work instructions (Task, Time allocated).

Step 3: Demonstrate how to create a sketch in SolidWorks software based on the given project.

Step 4: Ask trainees to create the sketch in SolidWorks software and monitor the procedures.

Step 5: Check whether the sketch is correctly created and provide clarification if any.

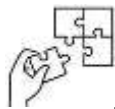
Step 6: Ask trainees to read key reading 2.2.2.

Step 7: Ask trainees to perform the application of learning 2.2.



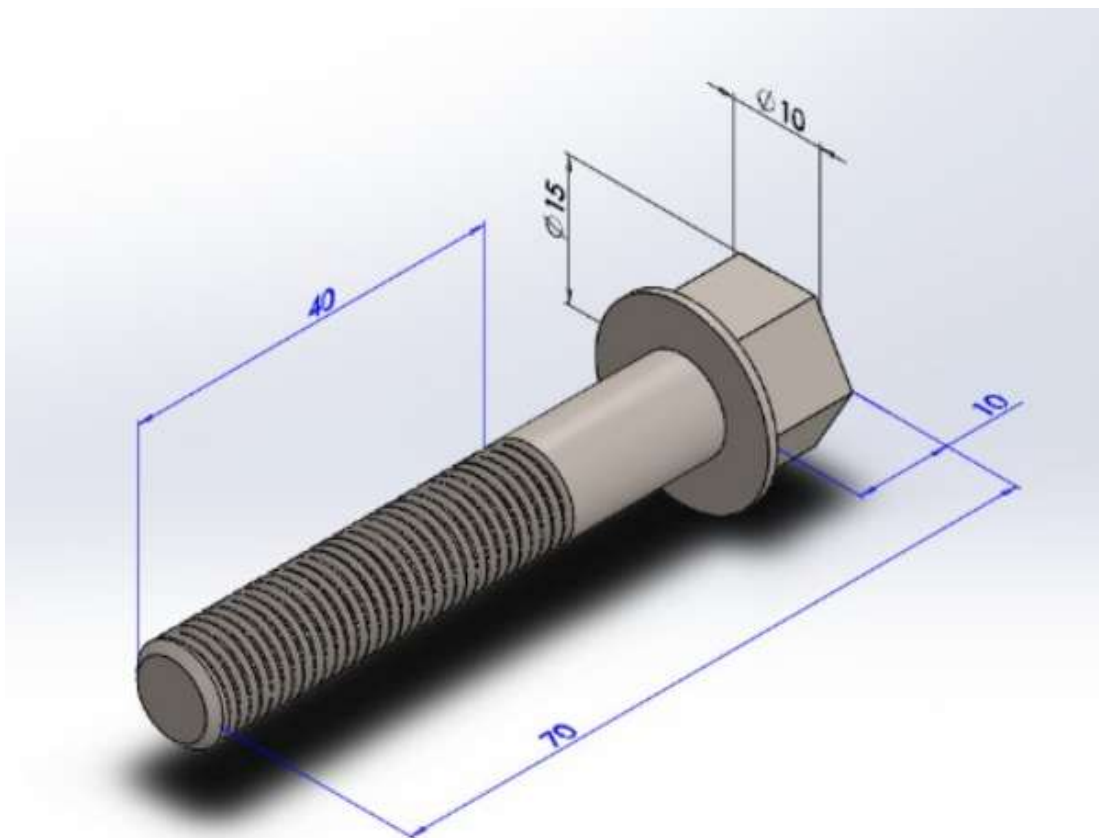
Points to Remember

- When creating sketches in SolidWorks, always start by selecting the correct plane and using precise dimensions to define the shape accurately. Use constraints like parallel, perpendicular, and coincident to maintain sketch integrity and ensure it is fully defined before proceeding to 3D modelling.



Application 2.2.

The school wants a ISO hexagonal headed bolt which will be used on one machine in manufacturing workshop. Ask your trainees to create the sketch of hexagonal headed bolt by referring to the bellow drawing.



Checklist

SN	Criteria	Indicators	Yes	No
1.	SolidWorks software is properly Lunched	1.1. Template is selected		
		1.2. Unit system(MMGS) is selected		
		1.3. Plane is selected		
2.	Sketch is well created	2.1. Sketch is created		
		2.2. Correct dimensions of the hexagon are applied		
		2.3. The hexagonal head is extruded to the required height		
		2.4. A circular sketch for the bolt shaft is created		
		2.5. Chamfers or fillets are applied as per the drawing		
		2.6. The bolt sketch is fully defined and complete		



Indicative content 2.3: Applying Part Features



Duration: 7 hrs



Theoretical Activity 2.3.1: Explanation of feature tools used in SolidWorks



Notes to the trainer:

- While delivering this activity, small groups can be used to explain the feature tools used in SolidWorks software
- For delivering this activity effectively trainer may use images, tutorial videos, and illustrations as didactic materials.



Key steps:

While delivering this activity, pass through the following steps:

Step 1: Introduce the activity and request trainees to answer the following question:

- i. How can you explain the feature tools used in SolidWorks software?

Step 2: Ask the trainees to provide and write their answers on flipchart/paper.

Step 3: Engage trainees in presentation of their findings

Step 4: Provide expert view using didactic materials and address questions or Concerns

Step 5: Ask trainees to read the key reading 2.3.1. In trainee manual.



Points to Remember

- The Extruded Boss feature allows you to create 3D solids from 2D sketches by extruding them in a specific direction,
- Extruded Cut feature removes material from a 3D part using a 2D sketch. The Revolved Boss and Revolved Cut features enable the creation or removal of material by revolving a sketch around an axis, ideal for rotationally symmetric parts. The Swept Boss feature creates complex shapes by sweeping a 2D profile along a defined path,
- The Lofted Boss feature connects multiple 2D profiles to create smooth, transitional 3D shapes



Practical Activity 2.3.2: Applying feature tools in SolidWorks software



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to apply feature tools in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

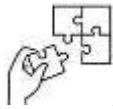
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to design a bolt then apply feature tools to the designed bolt.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to apply feature tools in SolidWorks software based on the given task.
- Step 4:** Ask trainees to apply feature tools in SolidWorks software and monitor the procedures.
- Step 5:** Check if the features are correctly applied and provide clarification if any.
- Step 6:** Ask trainees to read key reading 2.3.2.
- Step 7:** Ask trainees to perform the application of learning.2.3.



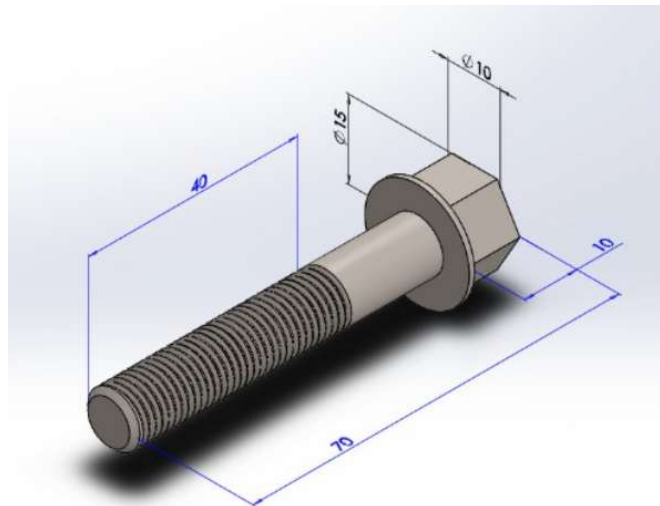
Points to Remember

- When applying feature tools in SolidWorks, always start with a well-defined sketch, as this forms the foundation for your 3D features. Ensure you understand the specific purpose of each feature tool, such as Extrude for adding material or Cut-Extrude for removing it. Pay attention to the parameters you set, like depth, angle, or radius, as these directly affect the final shape of your part. Preview the feature before finalizing to catch any potential issues early. Lastly, remember that features can be edited later



Application 2.3.

A manufacturing company requires ISO hexagonal-headed bolt for use in one of their workshop machines. Ask your trainees to apply the necessary features to design the hexagonal-headed bolt, using the drawing provided below as a reference.



Checklist:

SN	Criteria	Indicators	Yes	No
1.	Launching SolidWorks software is appropriately launched	1.1. Appropriate template is selected		
		1.2. Unit system(MMGS) is selected		
		1.3. Plane is selected		
		1.4. Appropriate Sketch is created		
2.	Applying features are correctly applied	2.1. Features are identified		
		2.2. Features are according to the work performed.		



Indicative content 2.4: Applying Visualizations



Duration: 5 hrs



Practical Activity 2.4.1: Applying various visualization in SolidWorks



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to apply visualisations in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to go in SolidWorks lab, apply different visualizations to designed hexagonal headed bolt using SolidWorks software.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to apply different visualization in SolidWorks software based on the given task.
- Step 4:** Ask trainees to apply different visualization in SolidWorks software and monitor the procedures.
- Step 5:** Check whether the visualisations are correctly applied and provide clarification if any.
- Step 6:** Ask trainees to read key reading 2.4.1.
- Step 7:** Ask trainees to perform the application of learning.2.4.



Points to Remember

- When applying visualization settings in SolidWorks, choose the appropriate display style for your design needs, carefully use appearances and Real View Graphics to enhance realism without overloading your system, and utilize section views and lighting adjustments to ensure accuracy and improve visual clarity



Application of learning 2.4.

The school want a hexagonal headed bolt to use on the milling machine. Ask your trainees to apply required visualisations to the designed hexagonal headed bolt.

Checklist

SN	Criteria	indicators	Yes	No
1.	SolidWorks software are properly prepared	1.1. Sketch is created		
		1.2. The bolt is created		
2.	Visualisations is correctly applied	2.1. Visualisations are identified		
		2.2. Visualisation is applied.		



Indicative content 2.5: Exporting Part



Duration: 5hrs



Practical Activity 2.5.1: Exporting part in SolidWorks software



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to export part in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

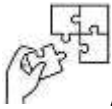
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to export designed bolt in PDF formats to the desktop of the computer.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to export your designs in SolidWorks formats
- Step 4:** Ask learners to export the designed projects and monitor the procedures.
- Step 5:** Check whether the created bolt is correctly exported in the correct format and provide clarification if any.
- Step 6:** Ask trainees to read key reading 2.5.1.
- Step 7:** Ask trainees to perform the application of learning.2.5.



Points to Remember

- The following are main steps for exporting part:
 - ✓ **Choose the Right Format:** Select the appropriate file format for your needs, such as **p**rt, **sldprt**, STEP, IGES, STL, and more **Set Export Options:** Adjust settings such as precision, resolution, or units in the export options to ensure the file meets your specifications and maintains model integrity.
 - ✓ **Save and Verify:** Save the exported file in the desired location and verify that it opens correctly in the target application or viewer to confirm the export was successful.



Application 2.5.

The school want a final document of designed hexagonal headed bolt. Ask your trainees to export the designed bolt in STL format then save it to the desktop of the computer.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Bolt is correctly created	1.1. SolidWorks software is successfully launched		
		1.2. Bolt is well created according to the instructions		
2.	SolidWorks part is properly exported	2.1. File location is properly identified		
		2.2. File STL format is correctly selected as required		
		2.3. Exported part is successfully and properly saved		



Learning outcome 2 end assessment

Written assessment

1. Read carefully the statement below and answer by using TRUE if the statement is correct or FALSE the statement is wrong:

- i. The Front Plane in SolidWorks is used as the reference for sketching features that are visible from the top of the part.

Answer: False

- ii. Custom planes in SolidWorks cannot be used for positioning components with respect to non-standard orientations.

Answer: False

- iii. Scenes and environments in SolidWorks can be configured to create a specific context or background for a part, which is useful for promotional materials

Answer: True

- iv. View orientation settings in SolidWorks allow users to only rotate and zoom but do not support panning to view different angles of a part

Answer: False

- v. Exporting a part in SolidWorks involves saving the 3D model in the native SolidWorks format to ensure compatibility with other software applications

Answer: False

- vi. A sketch is considered "Fully Defined" in SolidWorks when all its entities are constrained with dimensions or relations, and its position and size are completely determined

Answer: True

- vii. The Mirror feature in SolidWorks allows to creation of symmetrical parts by mirroring existing features or bodies across a specified plane.

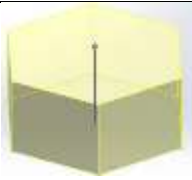
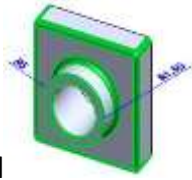
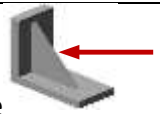
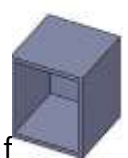
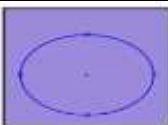
Answer: True

2. Complete the empty space by one of the following words: Sketch entities, full defined, under defined, over defined.

- i.**Sketch entities**..... are the basic geometric shapes that you create in a sketch. They include lines, arcs, circles, ellipses, and polygons
- ii. We say**Full defined**when all the geometric entities within the sketch have been fully constrained by dimensions and relations.
- iii. We say..... **Under defined**When there is some dimensions or geometric relationships that are not fully specified.

- iv. We say **Over defined** If the sketchhas redundant or conflicting dimensions or geometric relationships.

3. Match the sketch with the appropriate term in the table below.

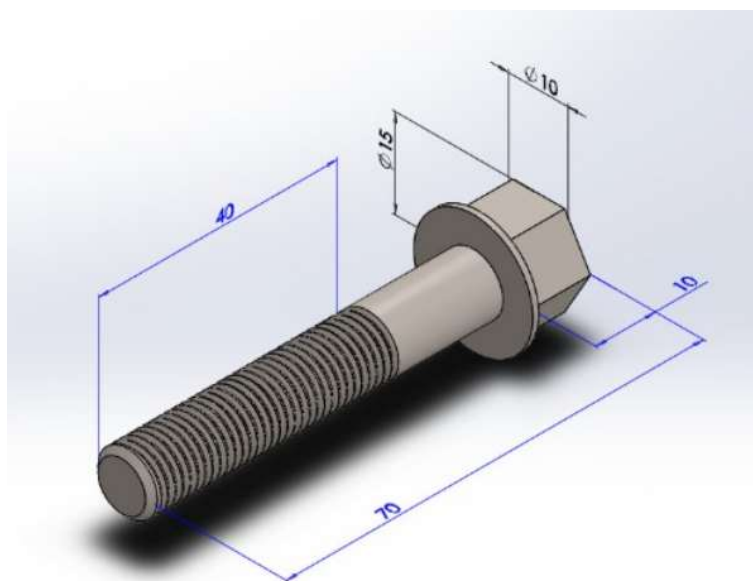
ANSWER	TERM	SKETCH
I.b	I. Revolved boss	a. 
II.a	II. Extruded boss	b. 
III.d	III. Fillet	c. 
IV.c	IV. Swept boss	d. 
V.f	V. Shell	e. 
VI.e	VI. Rib	f. 
VII.g	VII. Circular pattern	g. 
		h. 
		i. 
		j. 

Practical assessment

The school want a hexagonal headed bolts as shown on the figure below which will be used in manufacturing workshop.

Ask your trainees to perform the following tasks:

- ✓ Select the right plane to design an hexagonal headed bolts
- ✓ Create a fully defined sketch of a hexagonal headed bolts.
- ✓ Apply the appropriate features on the designed hexagonal headed bolts
- ✓ Apply visualisation on the designed hexagonal headed bolts
- ✓ Export it to the desktop of the computer in IGES format.



Checklist

SN	Criteria	indicators	Yes	No
1.	Plane is clearly Selected	1.1. Planes are identified		
		2.2. Selection of plane is done		
2.	Sketch is properly created according to the work to be done	2.1. Identification of sketch is done		
		2.2. Required sketch is created.		
		2.3. Sketch status checking is respected		

3.	Features are correctly applied	3.1. Features identification is done		
		3.2. All required features are applied		
4.	visualisation is correctly Applied	4.1. Identification of visualisation is done		
		4.2. Application of visualisation is performed		
5.	Part is correctly Exported	5.1. File location is selected		
		5.2. IGES File format is selected		
		5.3. Exported part is saved as requested		



Further information to the trainer

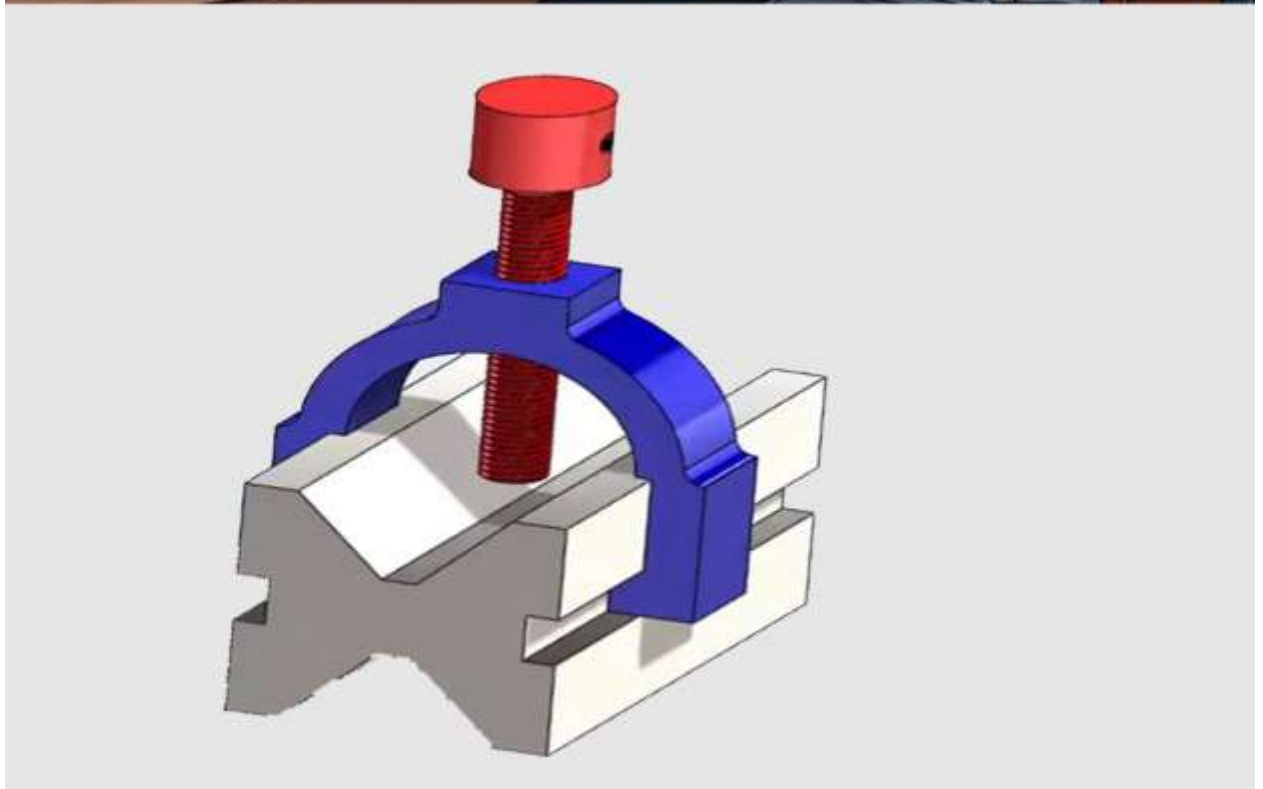
Cadartifex. (2021). A Power Guide for Beginners and Intermediate Users. SolidWorks.

Musto, W. H. (n.d.). Introduction to Solid Modeling Using SolidWorks.

Planchard, D. P. (n.d.). Engineering Design with SolidWorks.

Weber, G. V. (2021). Black Book. SolidWorks.

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 mates used in SolidWorks software. • Identification of different file locations. • Identification of different file formats.	• Importing components/inserting components. • Applying mates depending to the parts to be assembled. • Exporting assembly from SolidWorks according to the required format.	• Being persistent, team working, flexible, punctuality during project design • Having continuous learning and skill development • Being aware of a collaborative mind set for effective coordination with project stakeholders • Being proactive to problem-solving and troubleshooting abilities • Having a willing to experiment and explore new design possibilities • Being effective time management and organizational skills • Being effective time management and organizational skills



Duration: 25 hrs

Learning outcome 3 Objectives:



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

1. Import correctly the components used in SolidWorks design.
2. Describe properly type of mates used in SolidWorks design.
3. Apply correctly mates as used on the assembled SolidWorks components
4. Identify clearly the file locations used in solid works software
5. Identify clearly file formats used in SolidWorks software.
6. Export properly assembly used in SolidWorks design.



Resources

Equipment	Tools	Materials
• Computer • Projector	• SolidWorks software, • SolidWorks tutorials, • Online Courses, • Flash disk drive, • Hard disc drive, • Internet cable, • internet connections, • CD drive • Mouse	• Student workbook, • Papers, • Textbooks • SolidWorks Documentation, • Exercise Files • Sample Models, • Whiteboard • Visual Aids



Advance Preparation:

Before delivering this learning outcome, you are recommended to:

- Have a computer(s) which is/are in good condition
- Have computer(s) within installed SolidWorks software
- Prepare all SolidWorks software lab ready to be used
- Make sure SolidWorks software runs as intended
- Have a good and working projector



Indicative content 3.1: Importing Components/Inserting Components.



Duration: 8 hrs



Practical Activity 3.1.1: Importing components/inserting components in SolidWorks software.



Notes to the trainer

- Facilitation of this activity, trainer can demonstrate individual trainee how to Import components/insert components in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

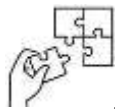
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to import or insert components of designed bolt and nut with washer in SolidWorks software
- Step 2:** Explain the task and provide clear work instruction (Task, Time allocated)
- Step 3:** Demonstrate how to importing and inserting components in SolidWorks design properly.
- Step 4:** Asks trainees to importing and inserting components in SolidWorks design and monitor the procedures.
- Step 5:** Checks whether importing or inserting components procedures are done correctly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 3.1.1. In trainee manual.
- Step 7:** Ask trainees to perform the task provided in application of learning 3.1.



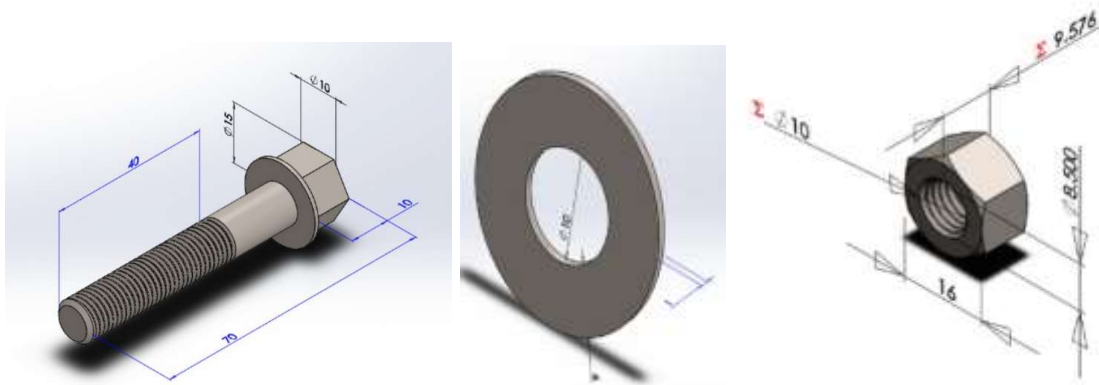
Points to Remember

- When inserting components in SolidWorks, always ensure that you are in Assembly mode and have selected the correct origin or reference point for placement. Use mating features like coincident, concentric, and tangent to precisely align and constrain the components, ensuring proper fit and functionality in the assembly.



Application of learning 3.1.

A technician has designed components of bolt and nut with washer and saved to the desktop of the computer as shown on the below drawing. Ask your trainees to import or insert the designed components to the SolidWorks Software workspace.



Checklist

SN	Criteria	Indicators	Yes	No
1.	Created components are correctly located	1.1. Template is selected.		
		1.2. The components to be inserted are located		
2.	Importing components or inserting components is performed properly	2.1. The files to imported are identified		
		2.2. The files are selected		
		2.3. Browsing files is done		
		2.4. Files are imported/ inserted		



Indicative content 3.2: Applying Mates



Duration: 12 hrs



Practical Activity 3.2.1: Applying mates in SolidWorks design



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to apply mates in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to apply mates to the imported or inserted components of bolt and nut with washer.
- Step 2:** Explain the task and provide clear work instruction (Task, Time allocated)
- Step 3:** Demonstrate how to apply mates in SolidWorks design properly.
- Step 4:** Asks trainees to apply mates and monitor the procedures.
- Step 5:** Checks whether application of mates is done properly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 3.2.1. in trainee manual.
- Step 7:** Ask trainees to perform the task provided in application of learning 3.2.



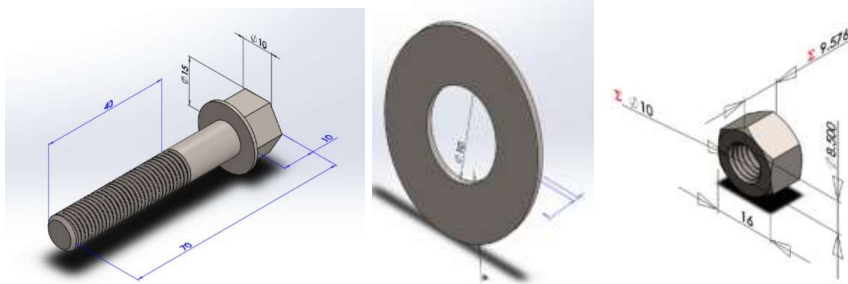
Points to Remember

- When applying mates in SolidWorks design, always start by selecting the appropriate type of mate (e.g., coincident, parallel, concentric) to accurately define the relationship between components. Ensure that each mate constraint is properly applied to avoid over-defining the assembly, and use the "Mate Alignment" option to adjust the orientation of parts if needed
- Pay attention to dependencies between mates. Editing or deleting one mate may affect others.



Application of learning 3.2.

In school there is a task of applying mates to the hexagonal headed bolt and nut with washer. Ask your trainees to apply the appropriate mates by referring to the bellow drawing.



Checklist

SN	Criteria	Indicators	Yes	No
1.	Component is correctly created	1.1. Template is selected to create single part		
		1.2. The mentioned components are created		
2.	SolidWorks software mates are correctly applied	2.1. All part to be assembled are imported		
		2.2. SolidWorks software mates are identified		
		2.3. Mates are applied according to the requirements.		



Indicative content 3.3: Exporting Assembly



Duration: 5 hrs



Practical Activity 3.3.1: Exporting assembly component made in SolidWorks design



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to export an assembly in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail video tutorials as didactic materials.



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to export the assembled component of Bolt and nut with washer.
- Step 2:** Explain the task and provide clear work instruction (Task, Time allocated)
- Step 3:** Demonstrate how to export an assembly in SolidWorks design properly.
- Step 4:** Asks trainees to export an assembly in SolidWorks design and monitor the procedures.
- Step 5:** Checks whether exportation is performed properly and provide clarifications if any.
- Step 6:** Ask trainees to read key reading 3.3.1. in trainee manual.
- Step 7:** Ask trainees to perform the task provided in application of learning 3.3.



Points to Remember

- When exporting an assembly component in SolidWorks, always check that the entire assembly is fully defined and free of errors before exporting. Select the appropriate file format (such as STEP, IGES, or STL) that meets the requirements of the project, and ensure all parts and references are included in the export to maintain assembly integrity.



Application of learning 3.3.

The school want a final document of assembled hexagonal headed bolt with nut and washer. Ask your trainees to export the assembly in STL format then save it to the desktop of the computer.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Component are correctly assembled	1.1. Hexagonal headed bolt part is created		
		1.2. Nut part is created		
		1.3. Washer part is created		
		1.4. All required mates are applied		
2.	SolidWorks assembly are properly exported	2.1. File location is selected		
		2.2. STL File format is selected as required		
		2.3. Exported assembly is saved		



Learning outcome 3 end assessment

Theoretical assessment

1. Read carefully the statement below and answer by using TRUE if the statement is correct or FALSE the statement is wrong:
 - i. Standard mates are basic mates that are commonly used to define the position and orientation of components. **ANS: TRUE.**
 - ii. Advanced mates provide additional functionality beyond standard mates and They allow for more complex relationships and constraints between components. **ANS: TRUE.**
 - iii. Mechanical mates are specialized mates that simulate mechanical behaviour and interactions between components. **ANS: TRUE.**
 - iv. When exporting a file format in SolidWorks design, the following format include (STEP, IGES, Parasolid, ACIS, STL). **ANS: TRUE**
2. Complete the dashed space by the appropriate term to make the sentence meaningful: Make Assembly from Part, Mates, and Analysis mates.
 - I.**Make Assembly from Part**..... allows you to convert the part into an assembly and add other components to it.
 - II. When assembling parts in SolidWorks design,**mates**..... are used to define the relationships between components.
 - III. **Analysis mates**..... allows you to verify the relationships and constraints between components in an assembly, ensuring that everything functions as intended.
3. Observe carefully the table below and match letters of corresponding to the type of mate with the letter stand for the example of mate to find the right answer.

ANSWER	TYPE OF MATE	EXAMPLE OF MATE
I.b	i. Standard mates	a. Belt/Chain, Universal Joint, Gear, Cam-follower, Rack and pinion, Screw Mate
II.c	ii. Advanced mates	b. Coincident, Parallel, Perpendicular, Concentric, Distance, Tangent Mate

III.a	iii. Mechanical mates	c. Width mate, Limit mate, Path mate, Linear/Linear , Hinge Mate, Symmetric
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Practical assessment

The SolidWorks software team designed a hexagonal headed bolt and nut with a washer and saved them in my documents of the computer hard disc.

Ask your trainees to:

- ✓ Insert the designed components hexagonal headed bolt and nut with a washer.
- ✓ Apply mates by using SolidWorks software to the hexagonal headed bolt and nut with a washer.
- ✓ Export the assembly to save in STL file format.



Checklist

SN	Criteria	indicators	Yes	No
1.	Components are correctly inserted	1.1. Components to be imported/ inserted are identified.		
		1.2. importing/inserting components is done		
2.	SolidWorks software mates are appropriately Applied	2.1. mates are identified to assemble component		
		2.2. mates are applied to assemble component		
3.	Assembly is properly Exported	3.1. File location is selected		
		3.2. STL File format is selected		
		3.3. Exported assembly is saved.		



Further information to the trainer

Cadartifex. (2021). A Power Guide for Beginners and Intermediate Users. SolidWorks.

Musto, W. H. (n.d.). Introduction to Solid Modeling Using SolidWorks.

Planchard, D. P. (n.d.). Engineering Design with SolidWorks.

Weber, G. V. (2021). Black Book. SolidWorks.

The illustration shows a technician in a white lab coat sitting at a desk, working on a computer. The monitor displays a 3D model of the Rail Adapter GSP Montage. Below the illustration is a detailed 3D model of the Rail Adapter GSP Montage and a technical drawing table.

Modell		Status		Name		Bezeichnung		Material	
G236JR		30.07.2019				Rail Adapter		1	
Mater. ohne Verantwortung						Zustimmung		Blatt	
Autodesk Fusion 360						1		1	

Indicative contents

4.1. Setting drawing sheet layout

4.2. Editing the title block

4.3. Importing views

4.4. Applying Visualization

4.5. Exporting the drawing

Key Competencies for Learning Outcome 4: Produce Drawing

Knowledge	Skills	Attitudes
• Description of drawing sheet layout. • Identification of the title block information • Description of drawing views in SolidWorks software. • Description of visualisations applied in SolidWorks software • Description of scale in SolidWorks software • Identify file locations of the drawing in SolidWorks software • Identification of file formats of the drawing used in SolidWorks software	• Setting drawing sheet layout. • Editing title block in SolidWorks software. • Importing views in SolidWorks software. • Applying visualisation • Exporting drawing in SolidWorks software.	• Having continuous learning and skill development • Having attention to detail and precision in design and documentation • Having adapting to software updates and industry requirements • Being aware of collaborative mindset for effective coordination with project stakeholders • Having a willing to experiment and explore new design possibilities • Being effective time management and organizational skills • Having conduct of ethical and professional architectural practice



Duration: 15 hrs

Learning outcome 4 objectives:



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

1. Describe properly the drawing sheet layout used in SolidWorks design.
2. Describe properly drawing views used in SolidWorks software.
3. Identify clearly elements of the title block used in SolidWorks design
4. Describe clearly the scales used in SolidWorks software
5. Describe properly visualisations used in SolidWorks software.
6. Apply correctly the visualisation of drawing used in SolidWorks design
7. Set correctly the drawing sheet layout used in SolidWorks software
8. Edit correctly the title block in SolidWorks software
9. Import properly the drawing views in SolidWorks software.
10. Export properly the drawing in the required files



Resources

Equipment	Tools	Materials
• Computer • Projector	• SolidWorks software, • SolidWorks tutorials, • Online Courses, • Flash disk drive, • Hard disc drive, • Internet cable, • internet connections, • CD drive • CD drive • Mouse	• Student workbook, • Papers, • Textbooks • SolidWorks Documentation, • Exercise Files • Sample Models, • Whiteboard • Visual Aids



Advance Preparation:

Before delivering this learning outcome, you are recommended to:

- Have a computer(s) which is/are in good condition
- Have computer(s) within installed SolidWorks software
- Prepare all SolidWorks software lab ready to be used
- Make sure SolidWorks software runs as intended
- Have a good and working projector
- Check the availability of electricity.



Indicative content 4.1: Setting Drawing Sheet Layout



Duration: 3 hrs



Theoretical Activity 4.1.1: Description of SolidWorks software drawing sheet layout.



Notes to the trainer:

- While delivering this activity, small group can be used for describing drawing sheet layout used in SolidWorks software.
- The use of images, videos, and illustrations as didactic materials is required.



Key steps:

While delivering this activity, pass through the following steps:

Step 1: Introduce the activity and request trainees to answer the following questions in SolidWorks software:

- i. What do you understand by the following sheet size
 - a. Standard sheet size
 - b. Custom sheet size
- ii. How can you differentiate landscape sheet orientation from portrait sheet orientation?

Step 2: Ask trainees to provide and write their answers on flipchart/paper.

Step 3: Engage trainees in presentation of their findings and choose correct answers.

Step 4: Present Expert view using didactic Materials and address questions or Concerns

Step 5: Ask trainees to read the key reading 4.1.1. in trainee manual.



Points to Remember

- When setting up a drawing sheet layout in SolidWorks, ensure you select the appropriate sheet size and scale to match the part or assembly being documented. Organize views effectively by placing the main view prominently and adding projected, section, and detail views as needed, while also labeling each view clearly and adjusting the dimensions to maintain readability and accuracy



Practical Activity 4.1.2: Setting drawing sheet layout design



Notes to the trainer

- Facilitation of this activity may be individual based, referring to the previous activity 4.1.1. Ask trainees to set a sheet size of A2 in portrait sheet orientation.in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

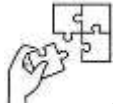
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to create a sheet size of A2 in portrait sheet orientation.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to set up sheet layout in SolidWorks design based on the given project.
- Step 4:** Ask trainees to set drawing sheet layout in SolidWorks software and monitor the procedures.
- Step 5:** Verify sheet layout created in SolidWorks if it is correctly created and provide clarification if any.
- Step 6:** Ask trainees to read key reading 4.1.2.
- Step 7:** Ask trainees to perform the application of learning.4.1.



Points to Remember

- In the SolidWorks software to create sheet layout you refer to the sheet size and the sheet orientation
- It is essential to respect the steps of setting up sheet layout to be successfully on the task



Application of learning 4.1.

The School want drawing sheet having the sheet with a width of 200 and a height of 600, in portrait orientation. Ask your trainees to setup the drawing sheet layout using SolidWorks software.

Checklist

SN	Criteria	Indicators	Yes	No
1.	SolidWorks software is properly launched	1.1. SolidWorks is launched		
		1.2. Drawing template is selected according to the work to be done.		
2.	Sheet layout is correctly set	2.1. Sheet size is set according to the design .		
		2.2. Sheet orientation is set according to the design.		



Indicative content 4.2: Editing the Title Block



Duration: 3 hrs



Practical Activity 4.2.1: Editing title block in SolidWorks software.



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to edit the title block in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

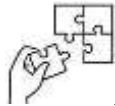
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the topic and ask trainees to edit the title block for five information.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to edit the title block in SolidWorks software based on the given task.
- Step 4:** Ask trainees to edit the title block in SolidWorks software and monitor the procedures.
- Step 5:** Check if the title block is properly edited and provide clarification if any.
- Step 6:** Ask trainees to read key reading 4.2.1
- Step 7:** Ask trainees to perform the application of learning.4.2.



Points to Remember

When editing the title block in SolidWorks, always right-click on the title block area and choose "Edit Sheet Format" to make changes. Use the text and line tools to modify the content, ensuring all necessary information such as part name, drawing number, and date is accurately updated before exiting the sheet format.



Application 4.2.

Manufacturing workshop need a technician for editing the title block for designed bolt and nut with washer. Ask your trainees to edit the title block by using SolidWorks software.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Sheet layout is properly set.	1.1. Sheet size is set		
		1.2. sheet orientation is selected		
2.	Title block is correctly edited according to the project	2.1. Title block table is created		
		2.2. The entire title block is edited according to the information to be filled.		



Indicative content 4.3: Importing Views



Duration: 3 hrs



Practical Activity 4.3.1: Importing drawing views in SolidWorks software.



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to importing views in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to create a drawing and import views by updating the views palette and access the model view.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to import views in SolidWorks software based on the given task.
- Step 4:** Ask trainees to import views in SolidWorks software and monitor the procedures.
- Step 5:** Check if the import of views is properly done and provide clarification if any.
- Step 6:** Ask trainees to read key reading 4.3.1
- Step 7:** Ask trainees to perform the application of learning.4.3.



Points to Remember

- It is important to keep view palette updated to ensure the correct views to be imported.
- It is essential to know procedures of importing views to achieve successfully and quickly the task.



Application 4.3.

In SolidWorks software Lab of your school, they need someone to import views of designed bolt and nut with washer by updating view palette and model view. Ask your trainees to import views by using SolidWorks software.

Checklist

SN	Criteria	Indicators	Yes	No
1.	Drawing sheet is well prepared	1.1. Template is selected		
		1.2. Drawing is created		
2.	views are properly Imported	2.1. View palette is updated		
		2.2. Model view is accessed		



Indicative content 4.4: Applying Visualization



Duration: 3 hrs



Practical Activity 4.4.1: Applying Visualization in drawing with SolidWorks software.



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to apply visualizations in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

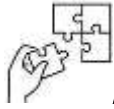
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to apply visualizations on the designed drawing of bolt.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to apply visualizations in SolidWorks software based on the given task.
- Step 4:** Ask trainees to apply visualizations in SolidWorks software and monitor the procedures.
- Step 5:** Check if the visualizations are properly applied and provide clarification if any.
- Step 6:** Ask trainees to read key reading 4.4.1
- Step 7:** Ask trainees to perform the application of learning.4.4



Points to Remember

- When applying visualization in SolidWorks, always assign appropriate materials and appearances to your model to reflect real-world properties accurately. Use scene settings, lighting, and camera angles to enhance the model's presentation, ensuring clarity and realism in your visualizations.



Application 4.4.

The school want a hexagonal headed bolt to use on the milling machine. Ask your trainees to apply required visualisations to the designed hexagonal headed bolt.

Checklist

SN	Criteria	indicators	Yes	No
1.	Drawing is properly produced	1.1. the sheet layout is set		
		1.2. Drawing is produced		
2.	Vsualisations is appropriately applied	2.1. Visualisations are identified		
		2.2. Visualisation is applied according to the desired work.		



Indicative content 4.5: Exporting the Drawings



Duration: 3 hrs



Practical Activity 4 5.1: Exporting the drawing in SolidWorks software.



Notes to the trainer

- Facilitation of this activity trainer can demonstrate individual trainee how to export the drawing by selecting appropriate file location and file format in SolidWorks software. For the effective delivery; it is recommended to:
 - ✓ Avail computers with an updated SolidWorks software installed.
 - ✓ Avail didactic materials such as video tutorials.



Key steps:

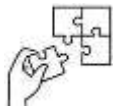
While delivering this activity, pass through the following steps:

- Step 1:** Introduce the activity and ask trainees to export the drawing by selecting appropriate file location and file format.
- Step 2:** Explain the task and provide clear work instructions (Task, Time allocated).
- Step 3:** Demonstrate how to export the drawing in SolidWorks software based on the given task.
- Step 4:** Ask trainees to export the drawing in SolidWorks software and monitor the procedures.
- Step 5:** Check if the export of the drawing is properly done and provide clarification if any.
- Step 6:** Ask trainees to read key reading 4.5.1
- Step 7:** Ask trainees to perform the application of learning.4.5.



Points to Remember

- When exporting a drawing in SolidWorks, ensure you select the correct file format (such as PDF, DXF, or DWG) based on the project's requirements. Always check the drawing scale, line weights, and annotations for accuracy, and verify that all views and dimensions are properly displayed before exporting



Application of learning 4.5.

The school want a final document of assembled hexagonal headed bolt with nut and washer drawing. Ask your trainees to export the assembly drawing in STL format.

Checklist

SN	Criteria	Indicators	Yes	No
1	Drawing is properly created	1.1. Drawing is created according to the instructions		
		1.2. All required settings are applied		
2.	Drawing is correctly exported	2.1. File location is selected		
		2.2. STL File format is selected as required		
		2.3. Exported part is saved		



Learning outcome 4 end assessment

Written assessment

1. Read carefully the statement below and answer by using TRUE if the statement is correct or FALSE the statement is wrong:

- i. In SolidWorks software, the sheet size refers to the dimensions of the drawing sheet you're working on, which is often used to create 2D representations of 3D models.

ANS: TRUE

- ii. The sheet orientation determines the alignment of the drawing sheet, whether it is in landscape (horizontal) or portrait (vertical) orientation.

ANS: TRUE

- iii. The view palette in SolidWorks provides a visual representation of the available views that can be imported into your drawing.

ANS: TRUE

- iv. Annotation in SolidWorks refers to adding dimensions, notes, symbols, and other graphical elements to the drawing to convey important information.

ANS: TRUE

- v. The display options in SolidWorks allow you to control the visibility and appearance of components and ensures that they effectively communicate the necessary information.

ANS: TRUE

- vi. The scale in SolidWorks determines the relationship between the size of the model and the size of the drawing on the sheet.

ANS: TRUE

- vii. Exporting drawings in SolidWorks is useful for sharing your work with others or for including it in reports and presentations

ANS: TRUE

2. Read carefully the following statements and choose the letter corresponding to the right answer.

- I. Among the following which elements may include in a title block.
- a) Annotation
 - b) File location
 - c) Drawing title

ANS: C)

- II. In the following elements which include in actions to be done while editing the title block table.
- a) View palette
 - b) Landscape
 - c) Merge cells

ANS: C)

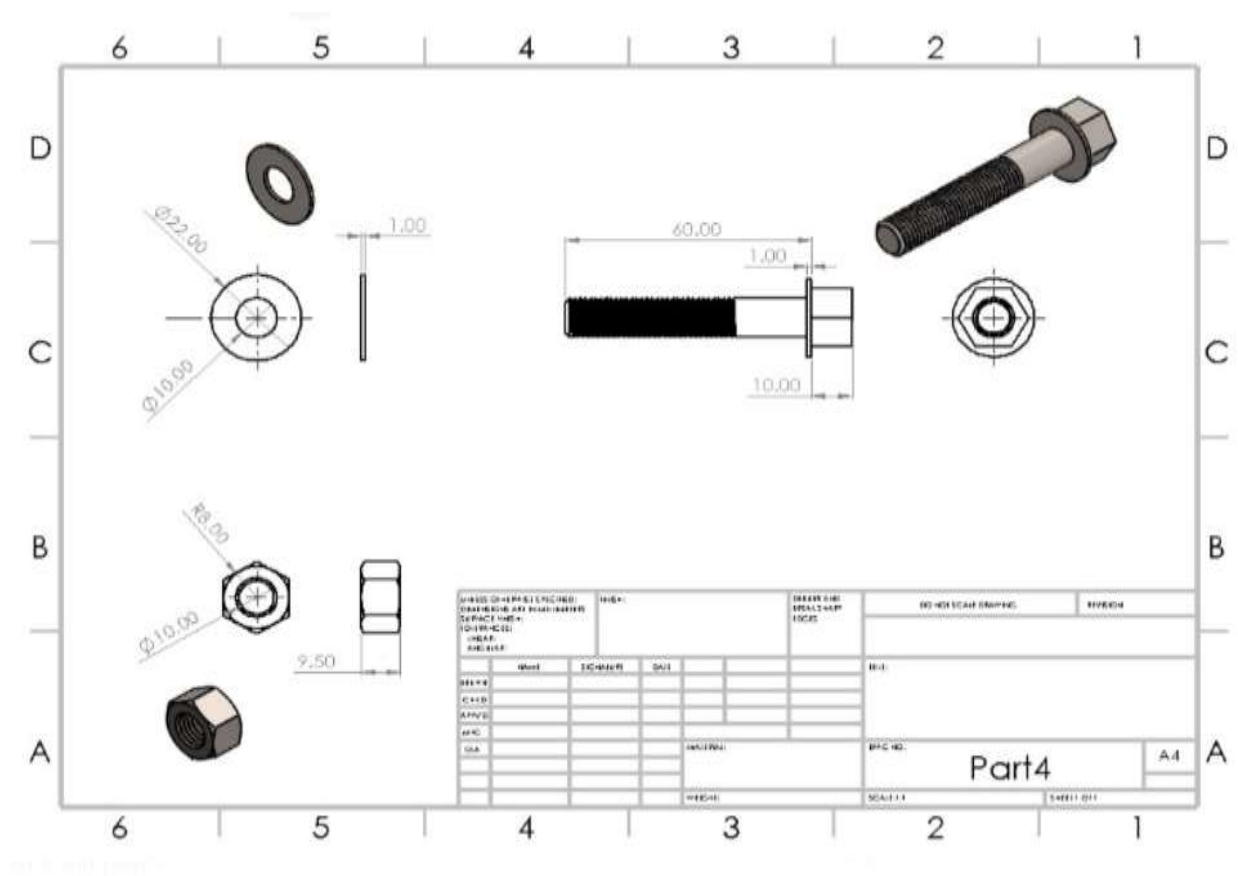
3. Complete the dashed space of the sentence by using one of the following terms of views with its definition to make sentence meaningful: Section views, Orthographic views, Isometric views.

- a)**Section views**..... allow you to cut away a portion of your model to show internal features or hidden details.
- b)**Orthographic views**..... show the model from different directions, such as front, top, right, left, bottom, and back.
- c)**Isometric views**.....display the model in a three-dimensional perspective, showing all three axes equally.

Practical assessment

The school want a final drawing of bolt and nut with washer shown by the figure below. Ask your trainees to:

- ✓ Set up sheet layout of A4 Landscape
- ✓ Edit the title block based on the project.
- ✓ Apply visualisation
- ✓ Export the final drawing



Checklist

SN	Criteria	indicators	Yes	No
1.	Setting drawing sheet layout is appropriately set	1.1. Sheet size is set		
		1.2. Sheet orientation is set		
2.	Title block is correctly edited	2.1. Title block table is created		
		2.2. Title block elements are inserted		
3.	Views are correctly imported	3.1. The views palette is updated		
		3.2. Model view is accessed		
4.	visualisation is correctly applied to the drawing	4.1. Different visualisations are identified.		
		4.2. visualisation is done		
5.	Drawing is correctly exported	5.1. File location is selected		
		5.2. File format is selected		
		5.3. Exported part is save as requested		



Further information to the trainer

Cadartifex. (2021). A Power Guide for Beginners and Intermediate Users. SolidWorks.

Musto, W. H. (n.d.). Introduction to Solid Modeling Using SolidWorks.

Planchard, D. P. (n.d.). Engineering Design with SolidWorks.

Weber, G. V. (2021). Black Book. SolidWorks.

https://help.solidworks.com/2021/english/Solidworks/sldworks/r_help.htm?rid

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

<https://www.goengineer.com/blog/apply-new-solidworks-drawing-sheet-format-futuredrawings>



October, 2024