

TRƯỜNG ĐẠI HỌC NÔNG LÂM TP. HỒ CHÍ MINH KHOA CƠ KHÍ – CÔNG NGHỆ





Assemblies Drawing



http://www2.hcmuaf.edu.vn/?ur=dangnh

Objectives

- ✓ Learn how to create assembly drawings.
- ✓ Learn how to create exploded assembly drawings.
- ✓ Learn how to create a parts list.
- ✓ Learn how to animate an assembly.
- ✓ Learn how to edit a title block

4.1. STARTING AN ASSEMBLY DRAWING

- Start a new drawing.
- Click the Assembly icon. Click OK.
- The Begin Assembly box will appear.
- Click the Browse . . . box. The Open box will appear.
- Click Block, Test, then click Open.The test block will appear on the screen.
- Click the Insert Components tool and insert a second block

TIP

To remove the fixed condition, locate the cursor on the (f) Block, Test <1> callout, right-click the mouse, and click the Float option. To return the block to the fixed condition or to fix another component, right-click the component name callout and select the Fix option.

4.2. MOVE COMPONENTS



4.3. ROTATE COMPONENTS



4.4. MATE (1)

- Click the Mate tool.
- Click the upper right edge of the second block inserted.
- Click the upper left edge on the first block.
- Click the OK check mark to clear the tools.
- Click the upper front surface of the second block.
- Click the upper front surface of the first block.



4.4. MATE (2)



4.4. MATE (4)



4.4. MATE (5)



4.5. CREATING AN EXPLODED ISOMETRIC ASSEMBLY DRAWING

- Click the Exploded View tool. See Figure 5-32.
- Click the top surface of the pin
- Move the cursor onto the Z-direction arrow and hold down the left mouse button. The arrow will turn yellow when selected.
- Drag the pin to a location above the assembly as shown.
- Click the OK check mark.
- Repeat the procedure and drag the Top away from the Bottom.
- Click the OK check mark.
- Save the assembly

4.6. CREATING AN EXPLODED ISOMETRIC DRAWING USING THE DRAWING FORMAT

- Create a new drawing using the Drawing format.
- Select the A-Portrait sheet format.
- Click the Browse ... box in the Model View Properties Manager.
- Select Block Assembly; click Open.
- Set the Orientation for Isometric and the Display Style for Hidden Lines Removed.

4.7. ASSEMBLY NUMBERS

- Click the Annotation tool.
- Click the AutoBalloon tool.



4.8. BILL OF MATERIALS (BOM OR PARTS LIST)

- To access the Bill of Materials tool click the Annotation toolbox, Tables, and Bill of Materials tools.
- Click the area of the Block Assembly
- Click the OK check mark



ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Block,Bottom		1
2	Block,Top		1
3	Ø15 Post	and the second second second	1

These are the parts' file.

4.9. ANIMATE COLLAPSE

- Open the Block Assembly.
- Right-click the Block Assembly heading in the Properties Manager.
- Click the Animate Collapse option.
- Click the Start button to stop the animation and return the assembly to the exploded position.
- Close the Animation Controller.

4.10. EDITING A PART WITHIN AN ASSEMBLY

- Right-click the Block, Top.
- Click the + sign to the left of the Cut-Extrusion 1 for the Block, Top heading in the Properties Manager.
- In this example the Cut-Extrusion 1 is the hole in the top block.
- Right-click the Sketch heading and click the Edit Sketch tool
- Double click the 01 5 hole value and enter a nev, value.
- Click the OK check mark.
- Click the Exit Sketch tool.
- Click the Edit Component tool.

4.11. TITLE BLOCKS (1)

- To Add Information to a Title Block
 - **1**. Click the Note tool located on the Annotation toolbar.
 - 2. Locate the Note box in the upper portion of the title block as shown.
 - 3. Type the company or school name.
 - 4. Click the check mark.
 - 5. Fill in other information as needed.

4.11. TITLE BLOCKS (2)

• To Change Fonts

- 1. Click the Tools heading at the top of the screen.
- 2. Click the Options . . . tool.
- **3. Click the Document Properties tab.**
- 4. Click the Annotations Font tool.
- 5. Click the Note tool.
- 6. Select the Times New Roman font.
- 7. Click OK and return to the drawing.



• Draw Details from page 230 to 232.



4.12. Project



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4.12. Project



4.12. Project



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Tạo các file: Assembly và Explored theo các hình sau:

01.. Assembly and Explore, Create a BOM

Project 5-3:



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02..Assembly and Explore, Create a BOM











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5. DRAWING ORTHOGRAPHIC VIEWS USING SOLIDWORKS

- Start a new drawing by click the New tool.
- Click the Drawing icon on the New SolidWorks Document box.
- Click OK.

The Sheet Format/Size box will appear. Accept the A-Landscape format

- Click OK.
- Click the X mark under the Model View heading.
- Move the cursor into the drawing area and rightclick the mouse.
- Select the Properties option.
- Click the Third angle button.
- Click OK.
- Click the Standard 3 View tool located among the Drawings tools

Cảm ơn các bạn đã theo dõi!

