PRACTICUM REPORT
ENGINEERING DRAWING
(19G04110401)

LILIS KRISDAYANA PUTRI
G041191004

AGRICULTURAL ENGINEERING PROGRAMME
DEPARTMENT OF AGRICULTURAL TECHNOLOGY
FACULTY OF AGRICULTURE
HASANUDDIN UNIVERSITY
MAKASSAR
2022

APPROVAL PAGE

TITLE   : ENGINEERING DRAWING PRACTICUM REPORT
NAME   : LILIS KRISDAYANA PUTRI
ID   : G041191004
GROUP   : I (ONE)

This Complete Report is Prepared as One of the Requirements to Complete the
Laboratory Examination for Technical Drawing Practicum Course
(19G04110401)

At

Agricultural Engineering Programme
Department of Agricultural Technology
Faculty of Agriculture
Hasanuddin University
Makassar
2022

Approved by,

Assistant 1   Assistant 2

MUH. RA'YAMSYAH   **MUSDALIFAH SUKMA**
NIM. G41116304   **NIM. G041181016**

Head of TSC   Coordinator Assistant

ALFIAN NURDIN   **A. MUH. FARHAN QIBRAN IBRAHIM**
NIM. G041181333   **NIM. G041181317**

Course Coordinator

MUHAMMAD RIZAL., S. TP , M.Si.

Makassar, December, 09 2022

PREFACE

Praise my gratitude to God Almighty, for His blessings and abundance of grace, so that the complete report of Engineering Drawing Practicum as one of the requirements for passing the Engineering Drawing Practicum course exam. In addition, the compiler also hopes that this report can add insight and knowledge for readers about Engineering Drawing.

The authors would like to express their deepest gratitude to all those who have helped me in completing this complete Engineering Drawing Practicum report, especially to all Assistants who have taught and guided me during the practicum. And do not forget to thank my friends who have helped me and taught me in this practicum. If there are still errors in the preparation of this complete report, then constructive criticism and suggestions will be accepted for the sake of the perfection of this report.

Makassar, December,9 2022

Lilis Krisdayana Putri

TABLE OF CONTENTS

GETTING TO KNOW SOLIDWORKS

3D MODELING

ASSEMBLY

DETAIL DRAWING

MINI PROJECT

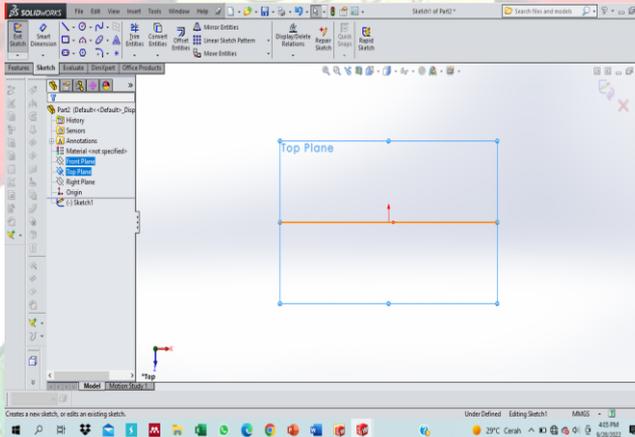# GETTING TO KNOW SOLIDWORKS

Wheel Making

1. Open SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
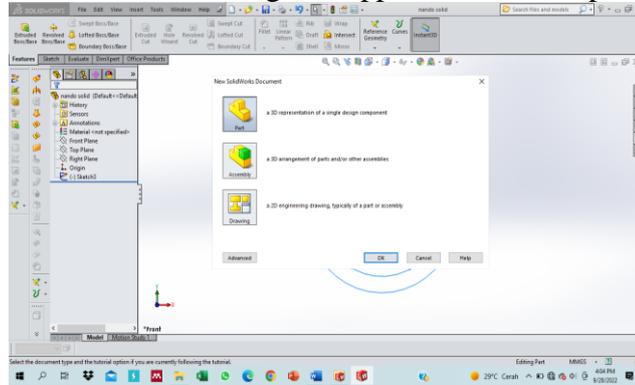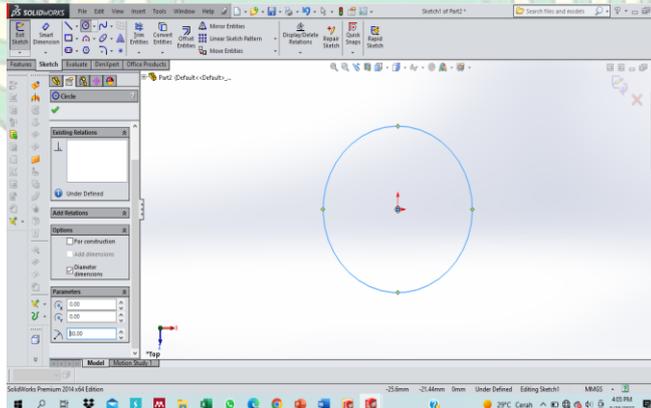

Picture 1. SolidWorks software view.

2. A workspace will appear to start drawing the part. To start drawing, select the top line then normal to.

Picture 2. Top plane worksheet view.

3. Create a circle by clicking circle then place the cursor on the datum axis as the center of the circle. Then set the size of the circle diameter to 60 cm.

Picture 3. Circle making view.

4. Do the same in procedure 3 and change the diameter of circle two to 60 mm and circle three to 40 cm.
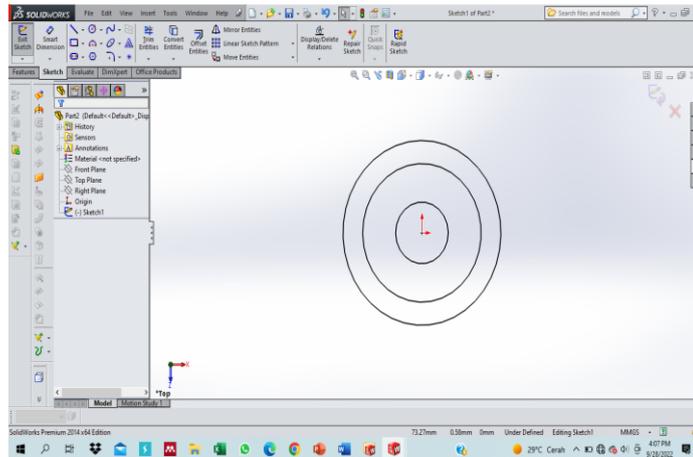
Picture 4. Circle diameter creation view.

5. Create a wheel by selecting the line menu then draw a line with a triangle shape connecting circle two and circle three.
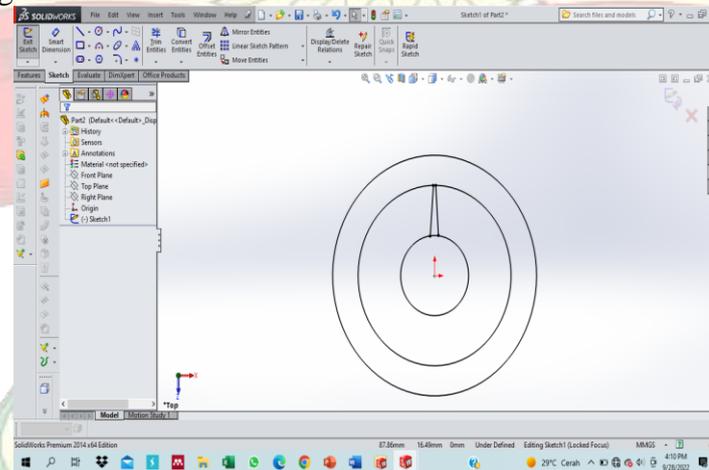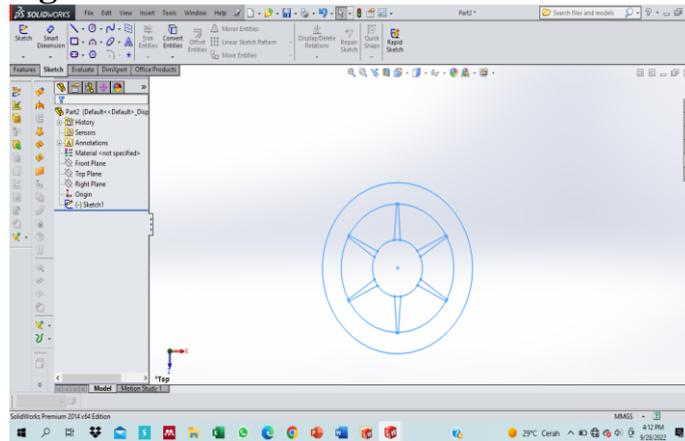

Picture 5. Line display.

6. Duplicate the line that has been created by clicking on the sketch pattern then multiply it by 6.
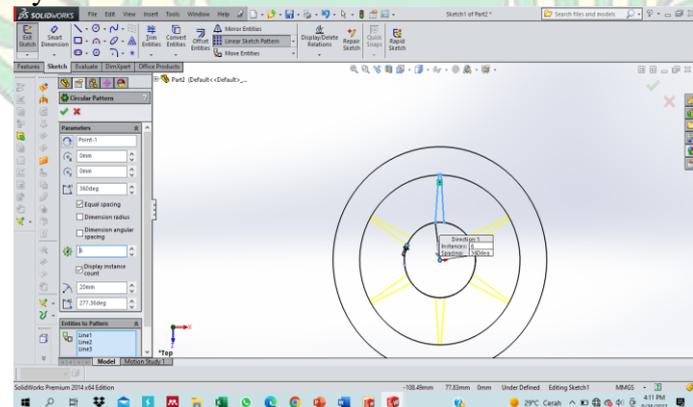
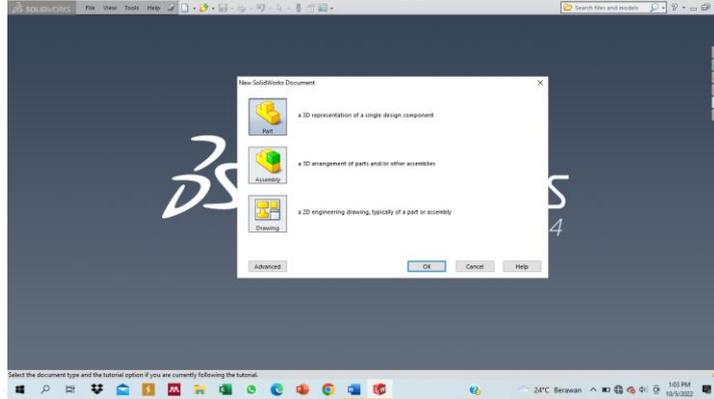Picture 6. Line doubling result display.

Wheel Making Results.

Picture 7. Wheel creation result display.

3D MODELING

Gear Making

1. Open SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
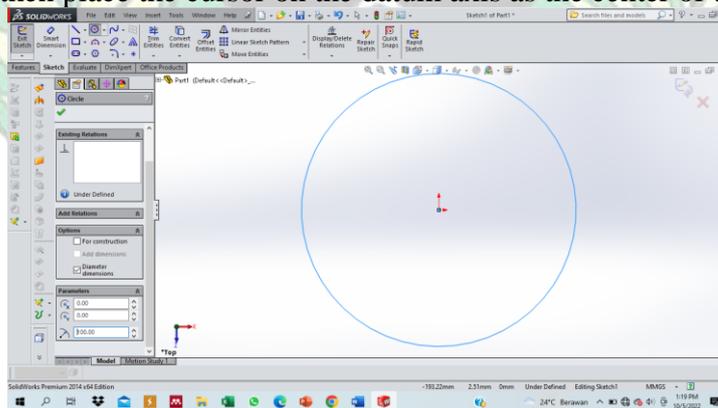

Picture 8. SolidWorks software view.

2. A workspace will appear to start drawing the part. To start drawing, select the top line then normal to then click sketch.
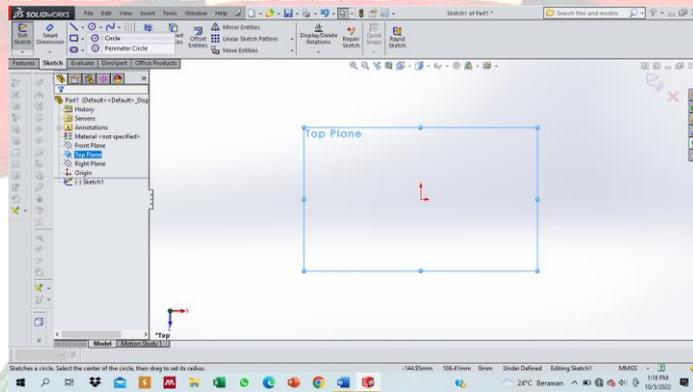

Picture 9. Top plane worksheet view.

3. Then create a circle by clicking the circle tool on the sketch with a size of 100 mm and then place the cursor on the datum axis as the center of the circle.
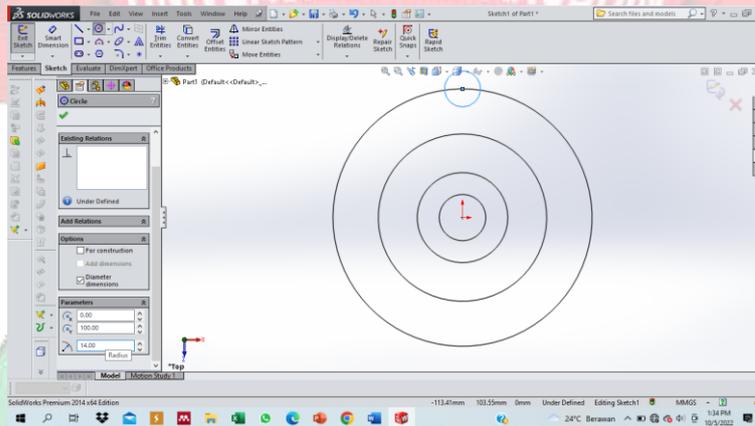

Picture 10. First circle view.

4. Create the second, third and fourth circle by clicking the circle tool on the sketch with sizes 65 mm, 35 mm and 18 mm then place the cursor on the datum axis as the center of the circle.
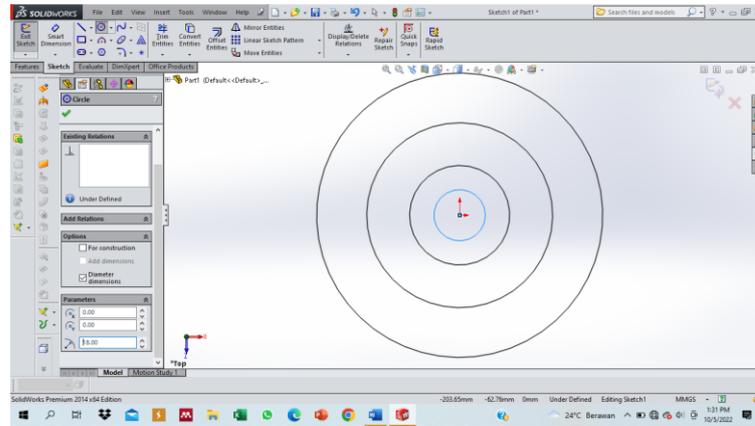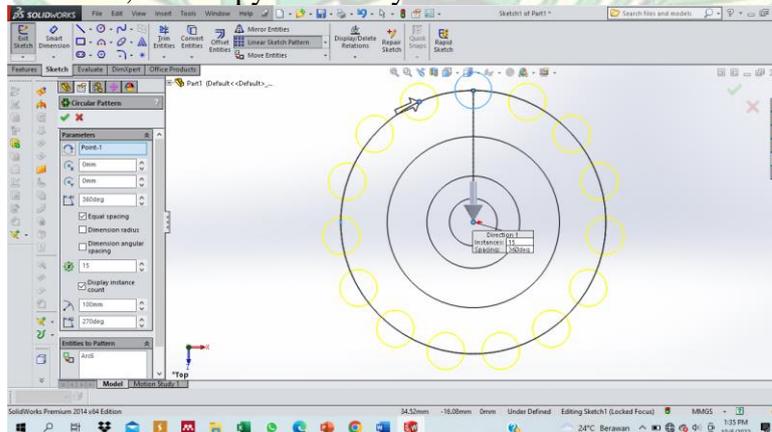

Picture 11. Second, third and fourth circle views.

5. The next step is to make a small circle with a size of 14 mm on the side of the first circle.
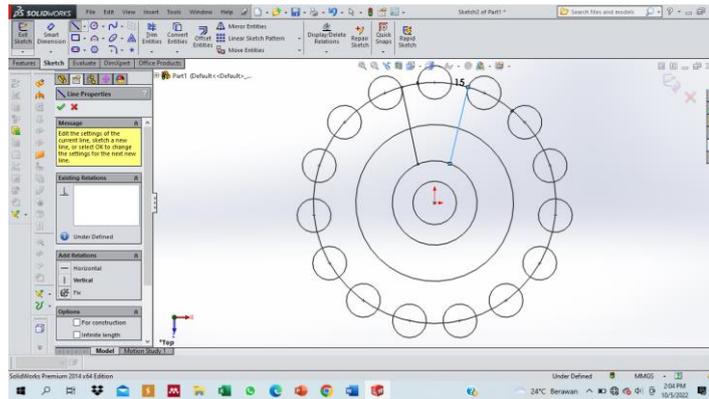

Picture 12. Small circle display.

6. Then select the circular sketch pattern tool then hover over the small circle that has been created, then copy it as many as 15 times.
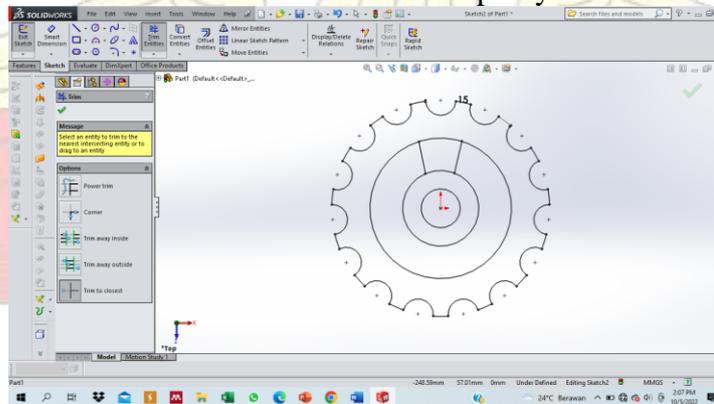

Picture 13. Small circle view after copying.

7. Create 2 straight lines connecting circles one, two and three by clicking the line tool.
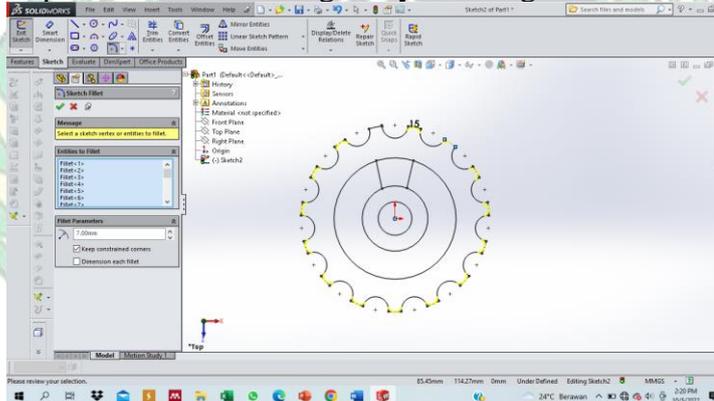

Picture 14. Straight line display.

8. Then select the trim etities tool to remove the part you want to remove.
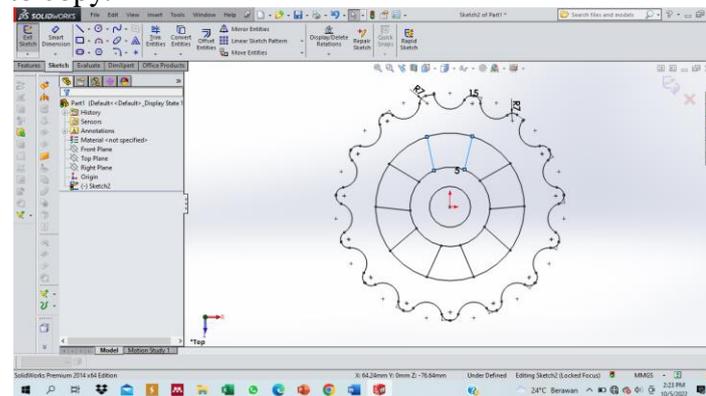

Picture 15. Result display of the removed part.

9. The next step is to fillet the outer gear by selecting the Fillet sketch tool.

Picture 16. Fillet sketch result display.

10. Then copy the straight lines that have been made previously as many as 5 each by selecting the circular sketch pattern tool then hovering over the straight line you want to copy.


Picture 17. Straight line result display after copying.

11. After that, delete the desired part by selecting the trim entities tool, then select the Fillet sketch tool to fillet each end of the line that has been deleted..
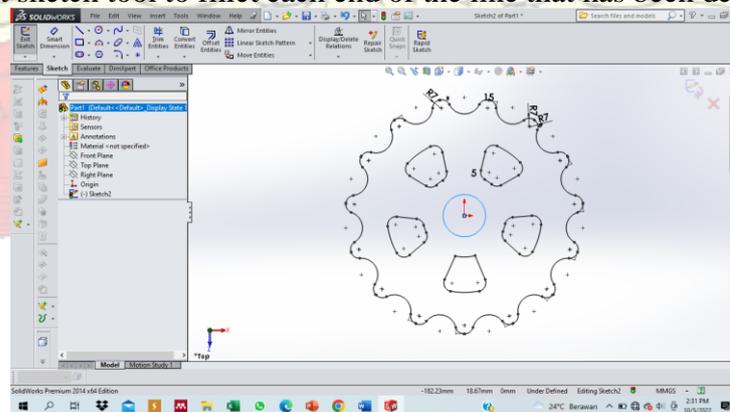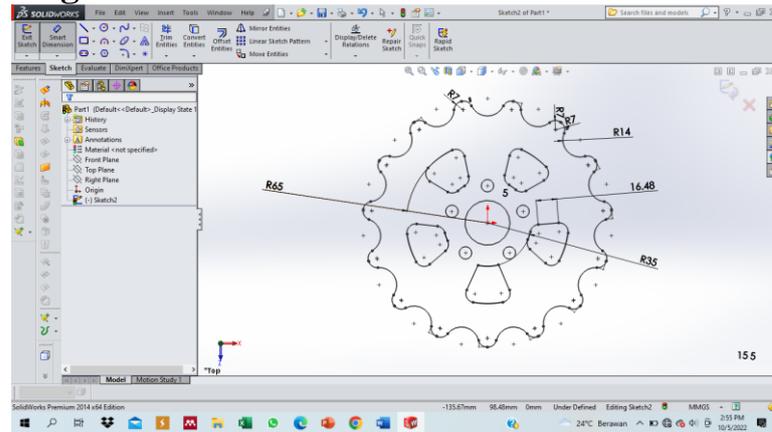

Picture 18. Trim and fillet result display.

12.   The next step is to create a small circle around the fourth circle by selecting the circle tool, then clicking the circular sketch pattern tool for the process of copying the small circle that has been made as many as 5 by hovering the cursor on the circle.
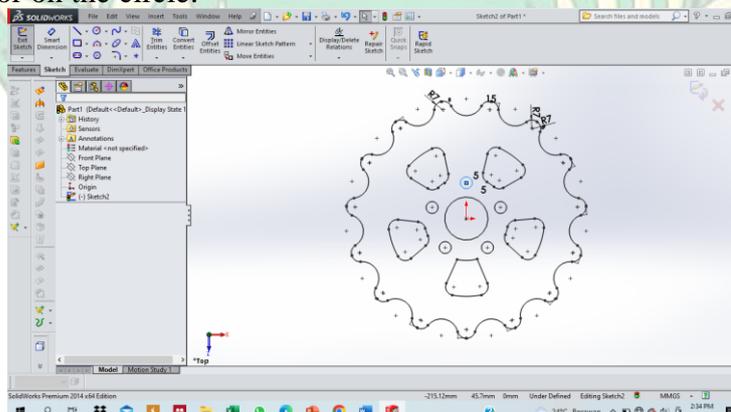
Picture 19. Small circle display.

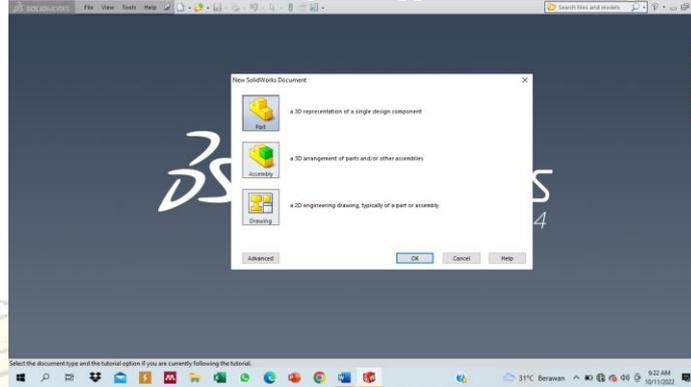# Gear Making Results

Picture 20. Display of gear Making results.
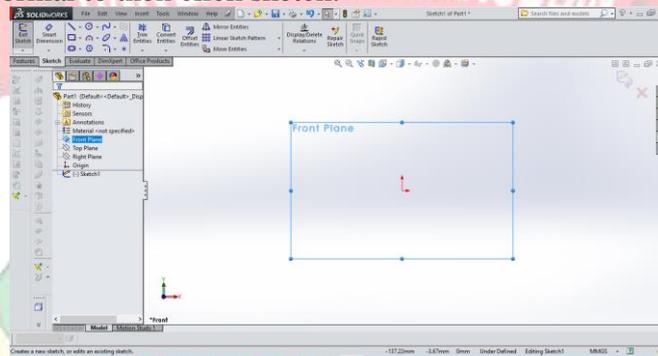
ASSEMBLY

Bolt and Nut Making

A. Bolt Making

1. Open SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
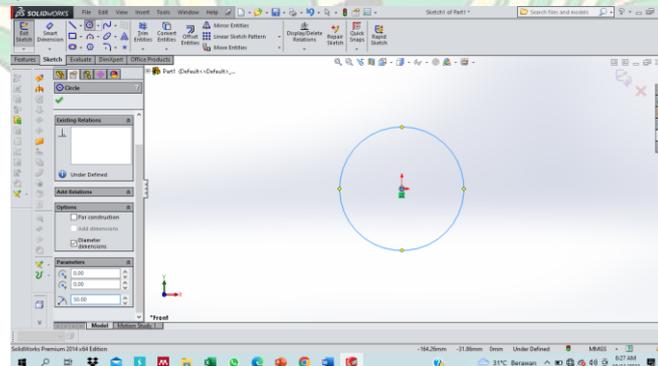

Picture 21. SolidWorks initial window display.

2. A workspace will appear to start drawing the part. To start drawing, select top line then normal to then click sketch.
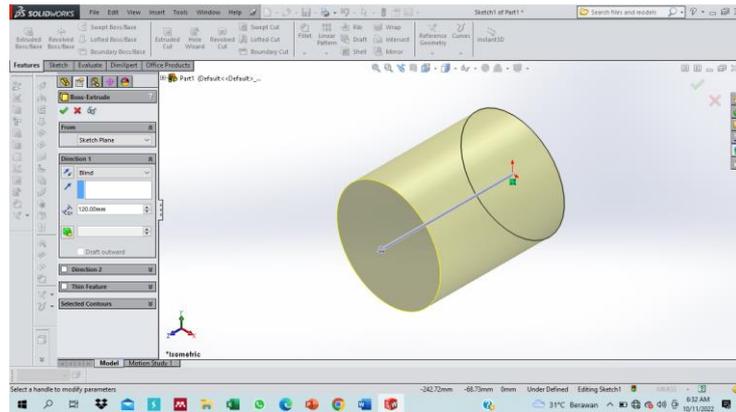

Picture 22. Top line view.

3. Create a circle by clicking circle then place the cursor on the datum axis as the middle of the circle. Then set the circle diameter size of 50 mm, then click the check mark.
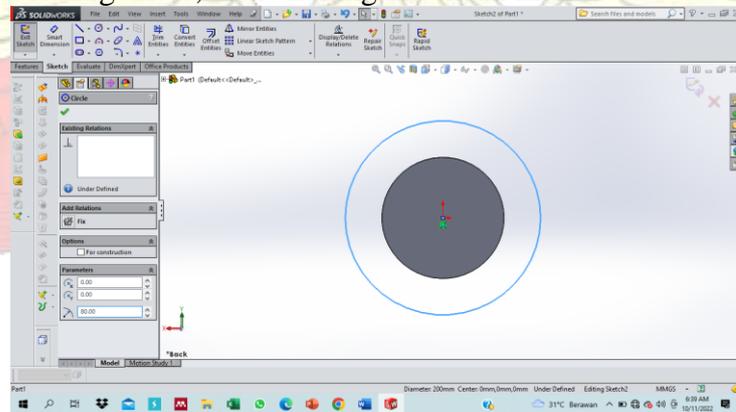

Picture 23. Circle making view.

4. Select extruded Boss/Bass on the features toolbar by giving a length of 120 mm, then click ok.
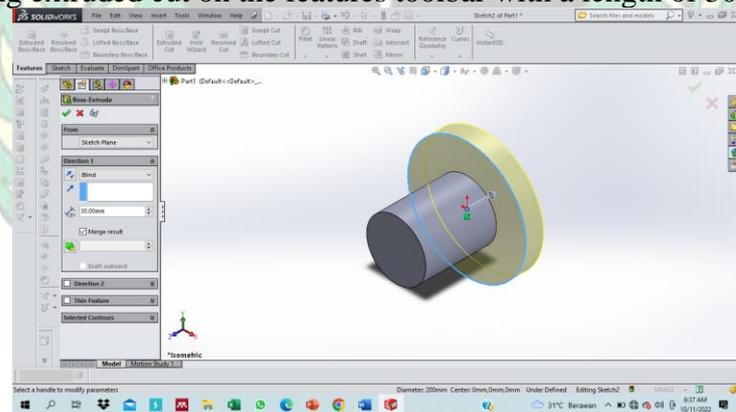

Picture 24. View of extruded boss/base.

5. The next step is to create a bolt head by making a circle with a diameter of 80 mm by selecting circle, then clicking the check mark.
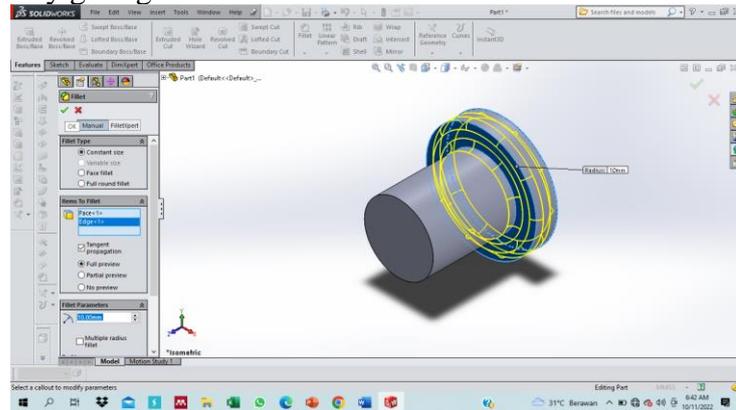

Picture 25. Bolt head Making display.

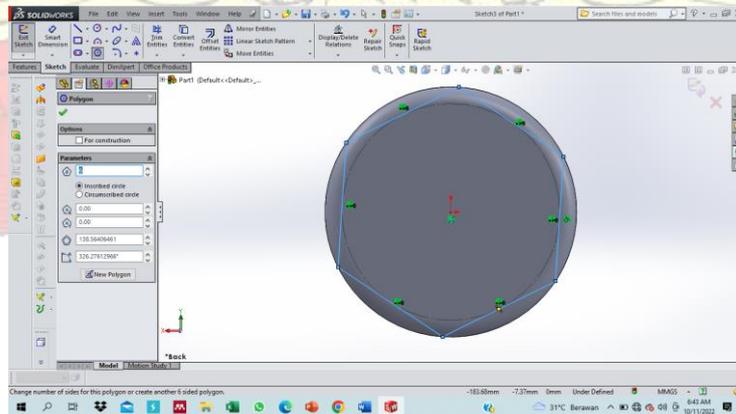6. Selecting extruded cut on the features toolbar with a length of 30 mm.

Picture 26. Bolt head Making display.

7. After that, select the fillet on the features and click the bottom and top of the bolt head by giving a diameter of 10 mm.
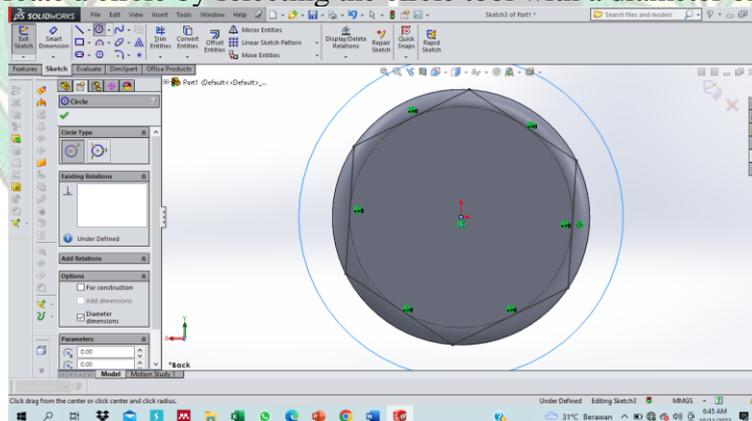

Picture 27. Fillet process view of the bolt head.

8. Then click the sketch toolbar and select the polygon tool to create a hexagon, then hover over the middle point of the bolt head, then click the check mark.
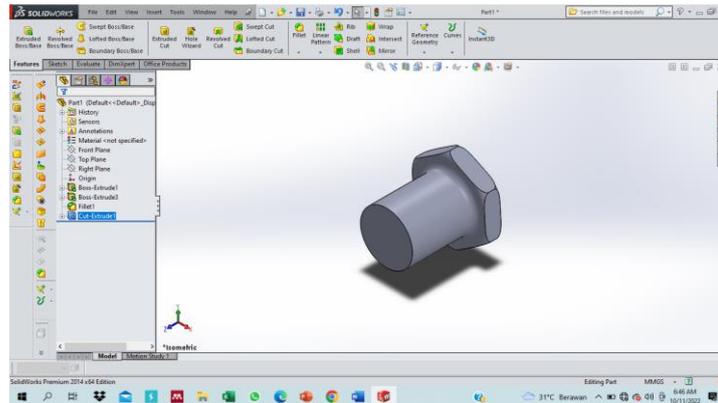

Picture 28. Hexagon creation view.

9. Next, create a circle by selecting the circle tool with a diameter of 100 mm.

Picture 29. Circle creation view.

10. After that, select extruded cut on the features toolbar by giving a length diameter of 30 mm.


Picture 30. Display after extruded cut.

11. The next step is to create a thread on the bolt by clicking on the object, then select appearances, then select face, then click the plus sign on the appearances display (colour), then select miscellaneous, then select pattern, then select the screw thread image, then set the thread distance, then click the check mark.
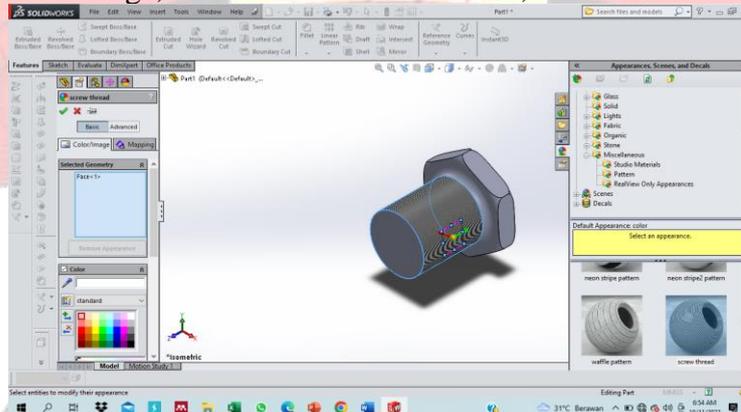
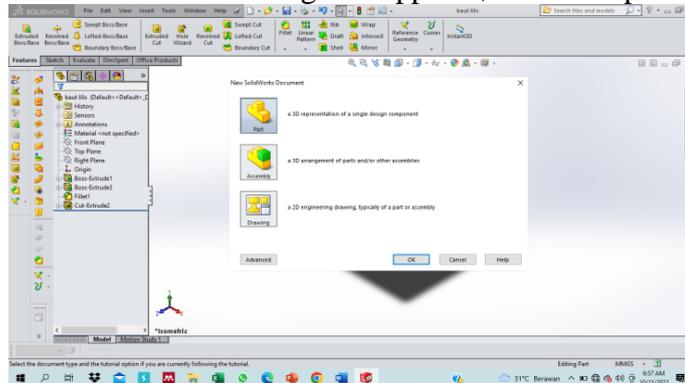Picture 31. Bolt Making result display.

12. Save the bolt worksheet by clicking the save command.
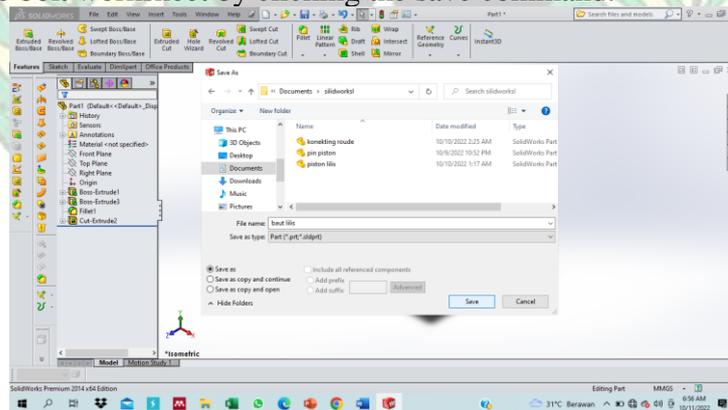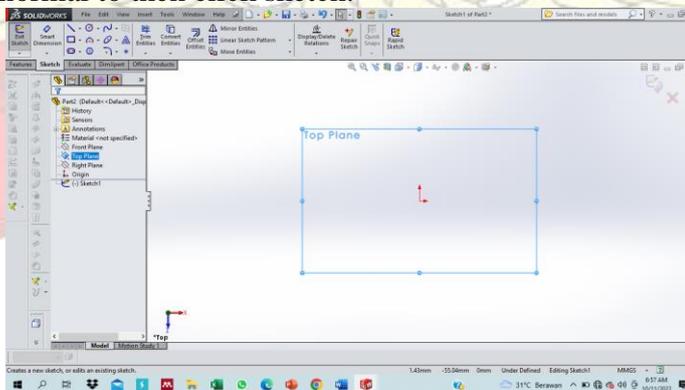

Picture 32. Worksheet save view.

B. Nut Making

1. Open SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
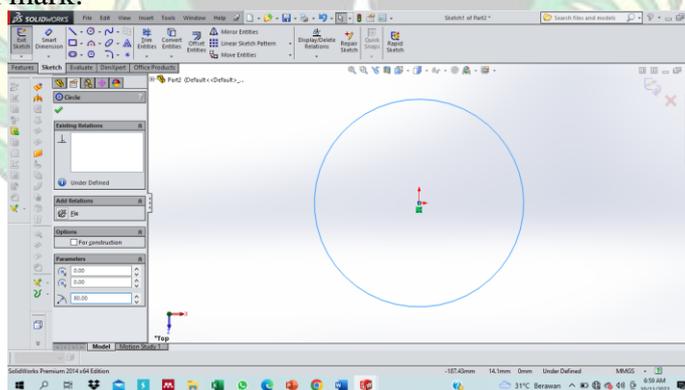

Picture 33. SolidWorks initial window display.

2. A workspace will appear to start drawing the part. To start drawing, select top line then normal to then click sketch.
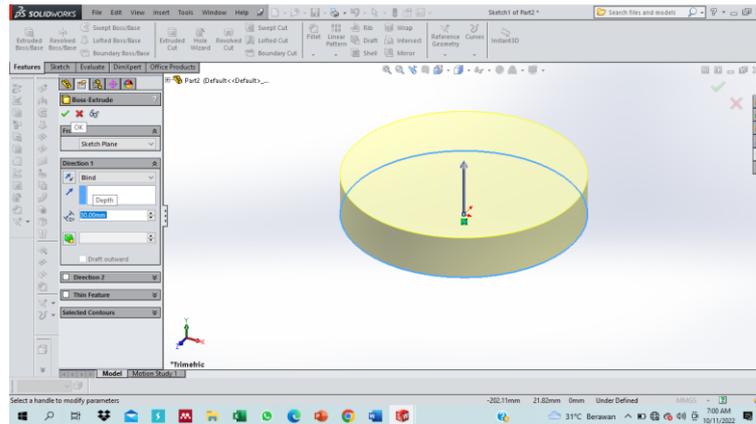

Picture 34. Top line view.

3. Create a circle by clicking circle then place the cursor on the datum axis as the middle of the circle. Then set the size of the circle diameter 80 mm, then click the check mark.
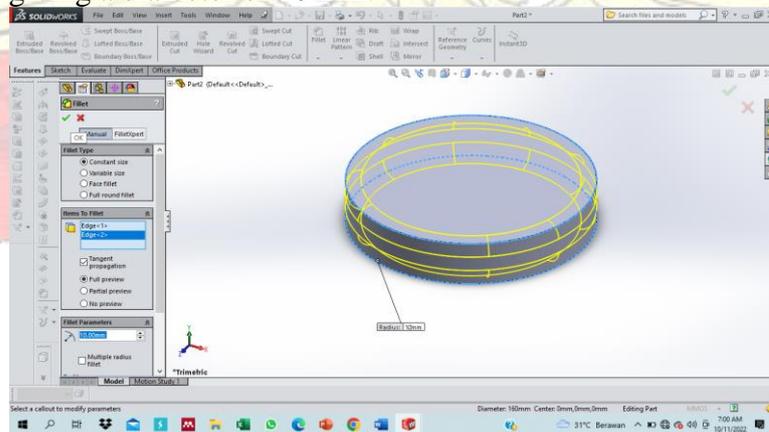

Picture 35. Circle making view.

13

4. Selecting extruded Boss/Bass on the features toolbar by giving a length of 30 mm.
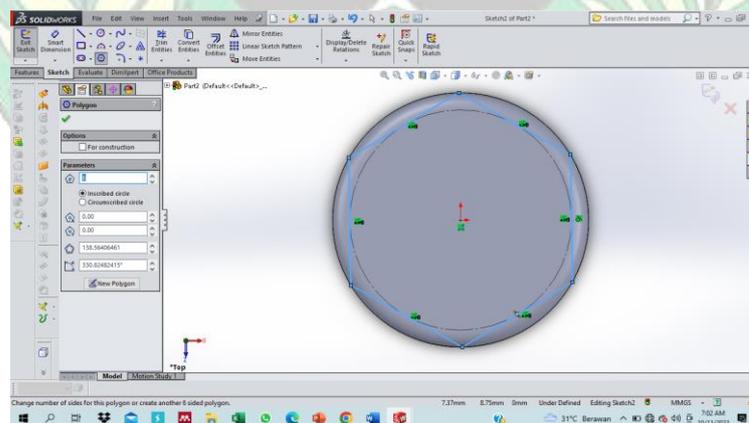

Picture 36. View of extruded boss/base.

5. After that, select the fillet on the features then click the bottom and top of the nut by giving a diameter of 10 mm.
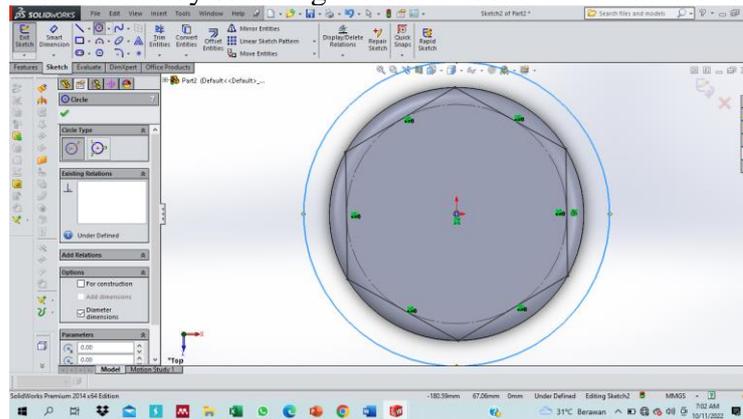

Picture 37. Fillet process view.

6. Then click the sketch toolbar and select the polygon tool to create a hexagon, then point the cursor at the middle point, then click the check mark.
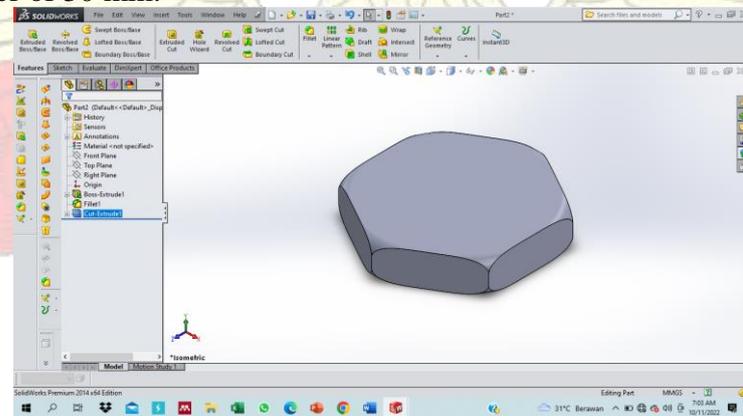

Picture 38. Hexagon creation view.

7. Next, create a circle by selecting the circle tool with a diameter of 100 mm.
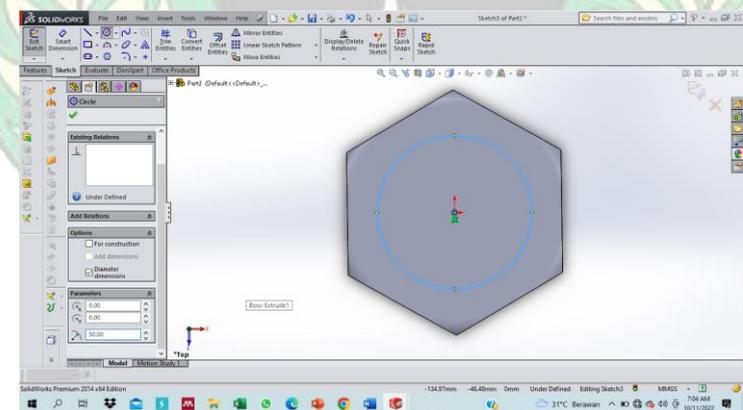

Picture 39. Circle creation view.

8. After that, select extruded cut on the features toolbar by giving a length diameter of 30 mm.
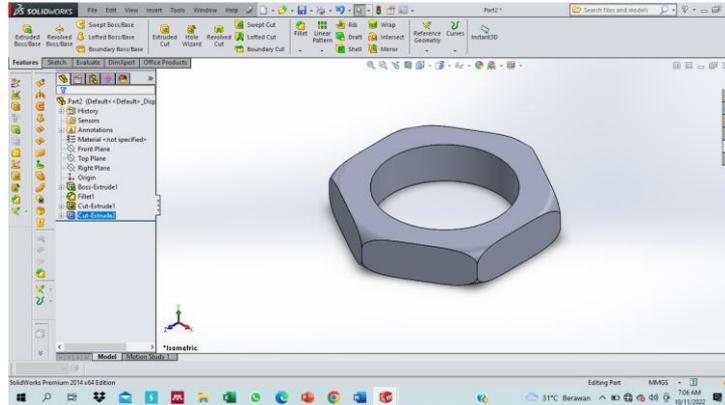

Picture 40. Result view of extruded cut.

9. Then make a hole in the nut by making a circle with a diameter of 50 mm at the middle point of the nut by selecting the circle tool.


Picture 41. Circle creation view.

10. After that, select extruded cut on the features toolbar by giving a diameter of 30 mm, then click the check mark.


Picture 42. Result display of hole making on the nut.

11. The next step is to create a thread on the nut by clicking on the object, then select appearances, then select face, then click the plus sign on the appearances (colour) display, then select miscellaneous, then select pattern, then select the screw thread image, then set the thread distance, then click the check mark.
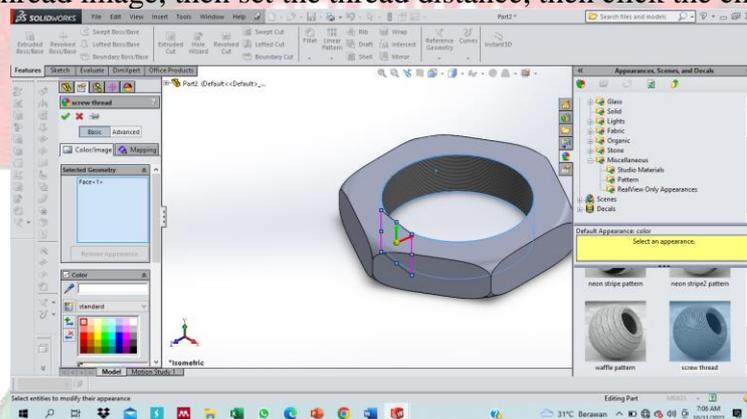
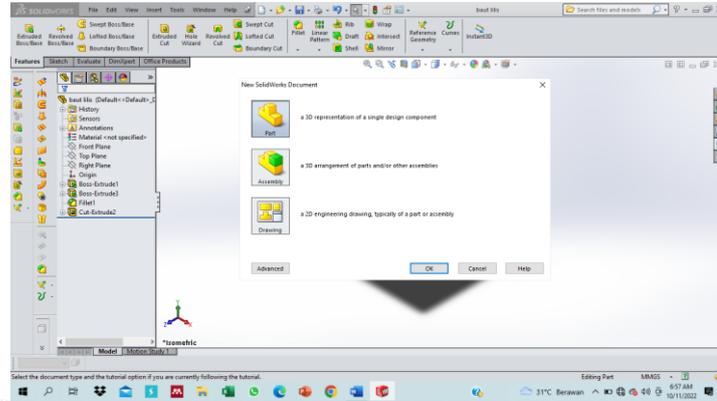Picture 43. Thread making display on the nut.

12. After that click the check mark and save the nut creation results.
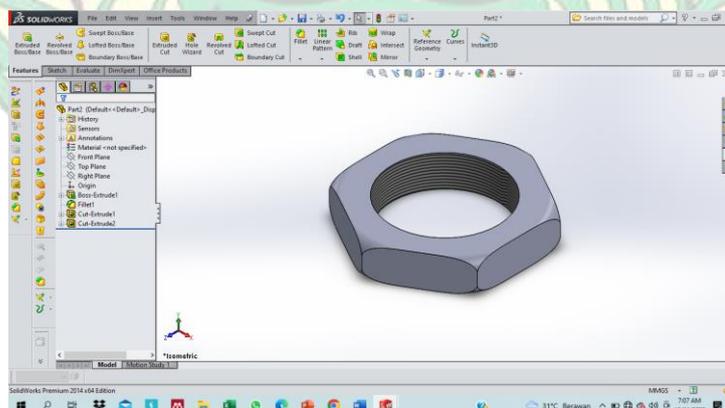
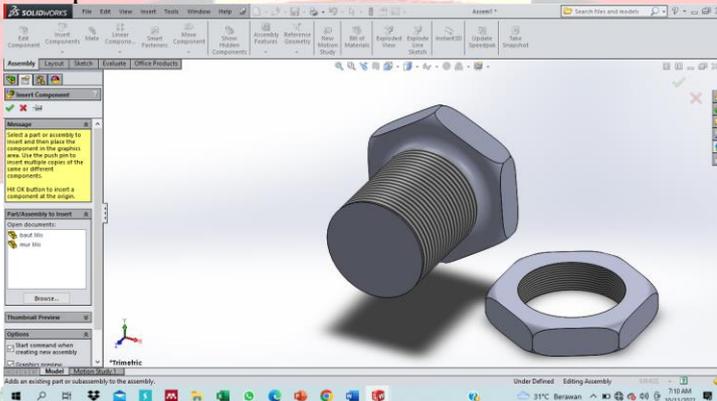Picture 44. Nut Making result display.

C. Nut and Bolt Assembly

1. Open the SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the Assembely menu then ok.
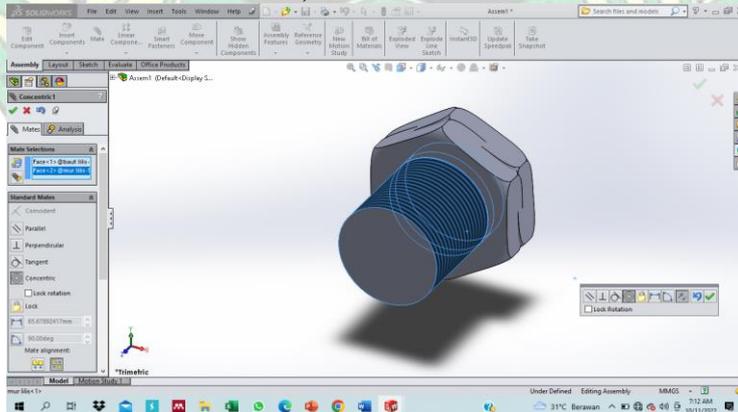

Picture 45. Work window selection view.

2. Insert the previously created nuts and bolts in the work window by selecting the insert components tool.
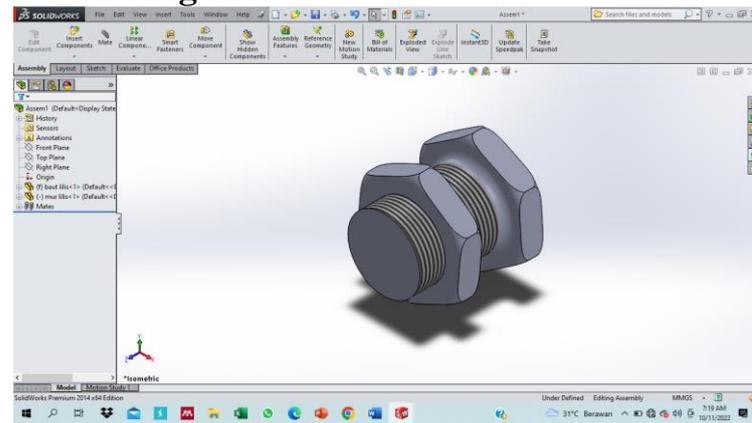

Picture 46. Display of nut and bolt components to be joined.

3. The next step is to click the thread on the bolt, then select the mate command, then click the thread on the nut, then click the check mark.

Picture 47. Nut and bolt mate result display.

Bolt and Nut Making Results

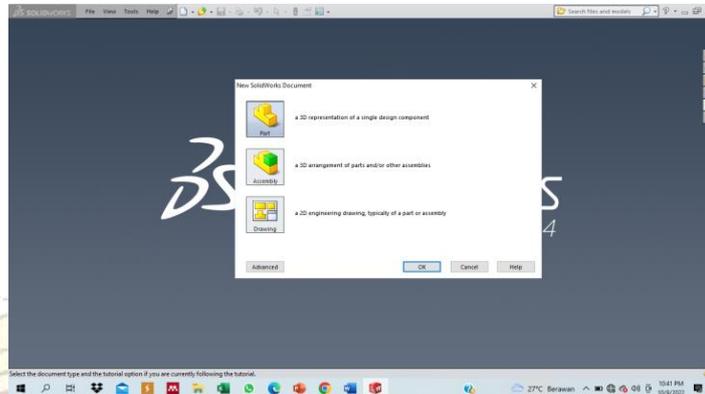Picture 48. View of the result of combining nuts and bolts.
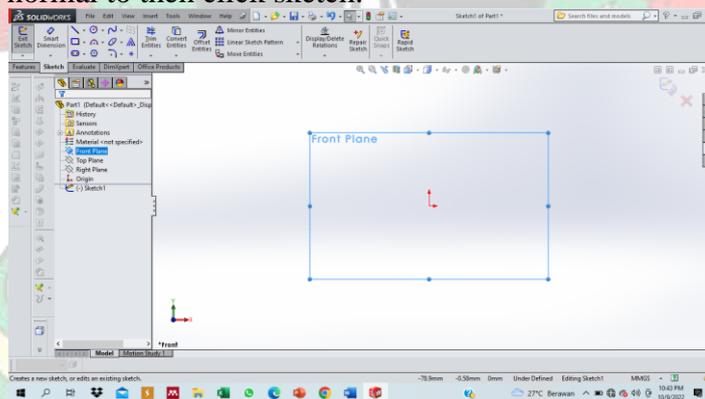
DETAIL DRAWING

Piston Making
A. PISTON PIN

13. Open the SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
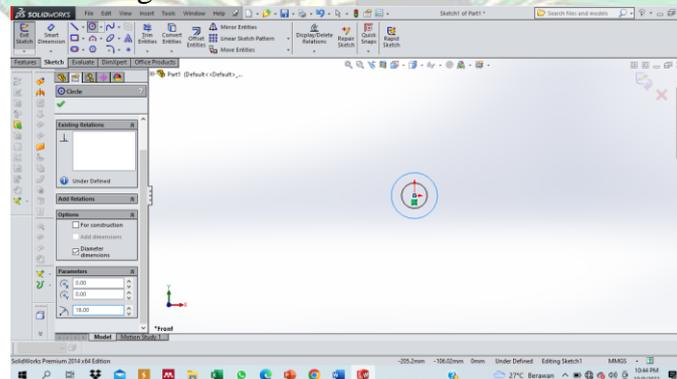

Picture 49. SolidWorks initial window display.

14. A work plane will appear to start drawing the part. To start drawing, select front line then normal to then click sketch.
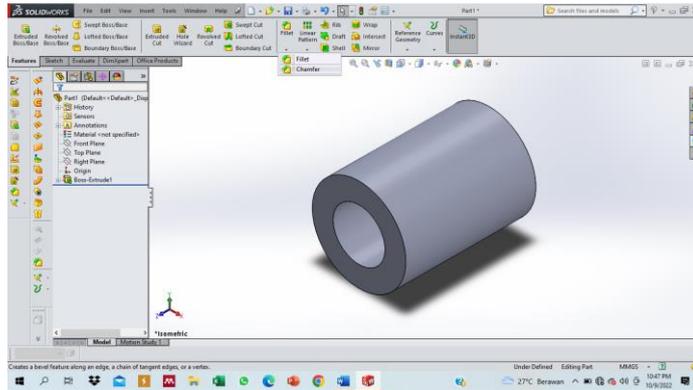

Picture 50. Front line view.

15. Create 2 circles using the circle tool with diameters of 10 and 18 mm.

Picture 51. Circle view.

16. Then select features then extruded boss/bass then select a diameter of 50 mm, then ok.

Picture 52. Result view of extrude boss/base.

17. Then create a chamfer by selecting fillet and then selecting chamfer. Select the first Chamfer type with a distant of 0.5 mm, then in the item to chamfer select the hole and the outside, then click ok.
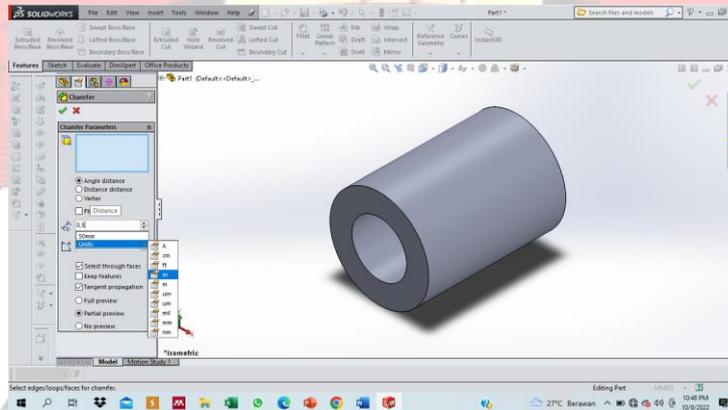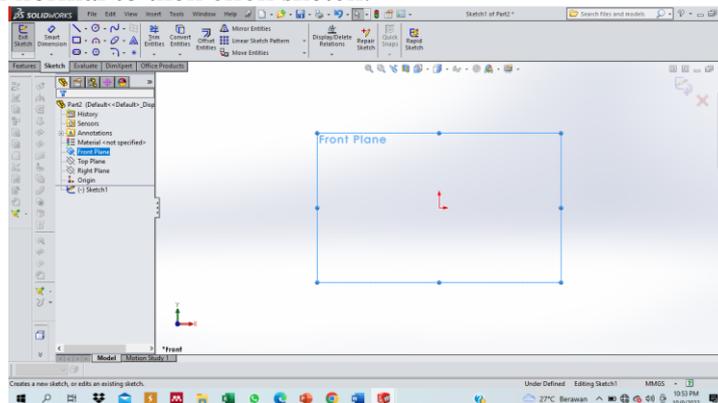

Picture 53. Chamfer command display.

18. Then save the piston pin.

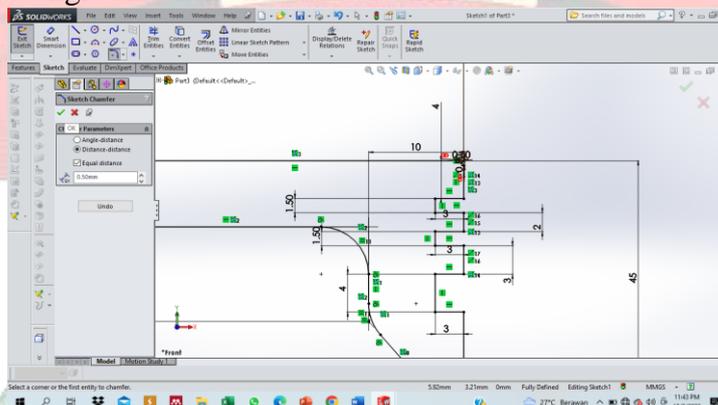
Picture 54. Piston pin save display.

B. PISTON HEAD

1. Select the file command then new, then select the part menu, then select front line then normal to then click sketch.
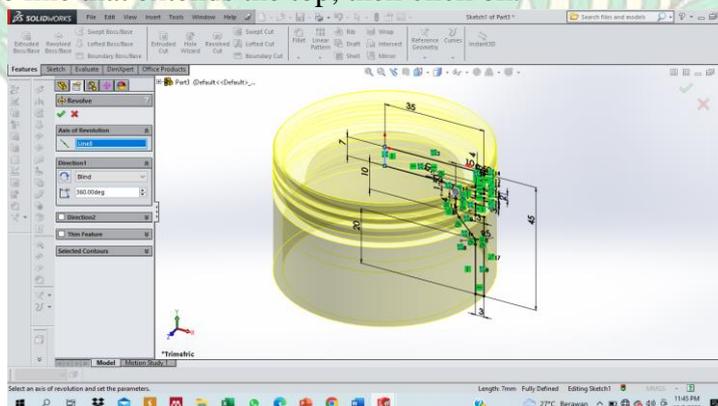

Picture 55. Front line view.

2. Create a series of piston heads by creating a pattern using the line, fillet, corner rectangle and trim entities tools then click ok.


Picture 56. Piston head pattern display.

3. After that, select features then click revolved boss/bass, for axis of revolution select the line that extends the top, then click ok.


Picture 57. Result view of revolved boss/base.

4. The next step is to click on the second circle from the outside then select convert entities then create a line with the line command at the middle of the bottom of the piston head, then select trim entities then cut the right circle line, then fillet the corner of the line, then click mirror entities then click the part you want to copy then click okay.
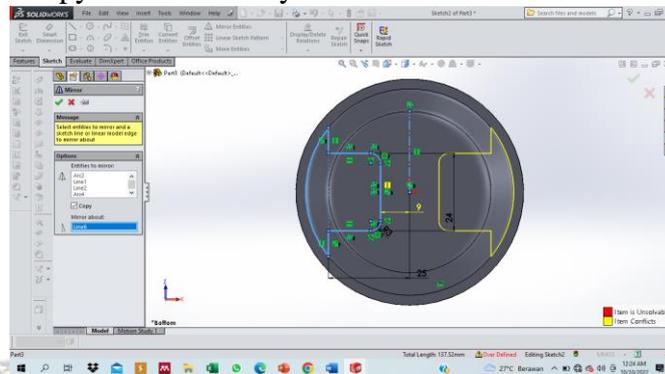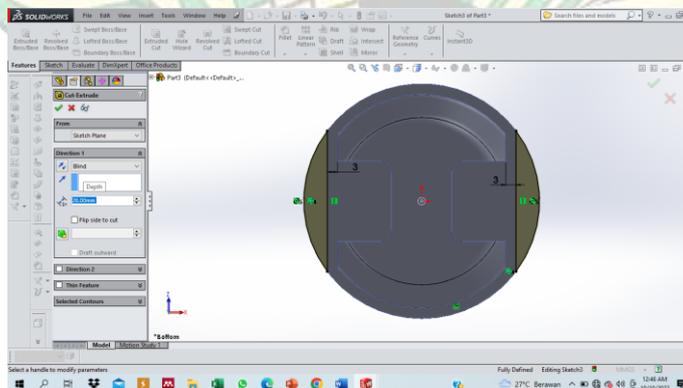

Picture 58. Bottom view of piston head.

5. After that, click the extruded boss/bass and change the return direction by clicking the direction arrow, then change the blind to up to next and click ok.
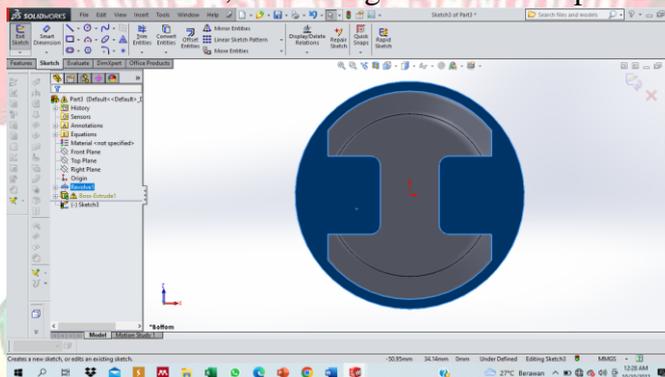

Picture 59. Result view extruded boss/base.

6. Create a line with the outer point of the circle using the line tool, then trim entities and cut the middle of the circle, then select extruded cut by giving dimensions 26mm then click ok.


Picture 60. Bottom view of the piston head.

7. Make a center line on the side of the piston by clicking the center point of the box, then make 2 circles with the center point as a reference with a small circle diameter of 18 mm, then make a line with the line command then trim entities and cut the large circle that has been made, then fillet each corner of the line, then click mirror entities and click the part you want to copy, then click ok. Next, select extruded cut then select selected contours then select all the parts that have been made before, then change the blind to trough all then click ok.
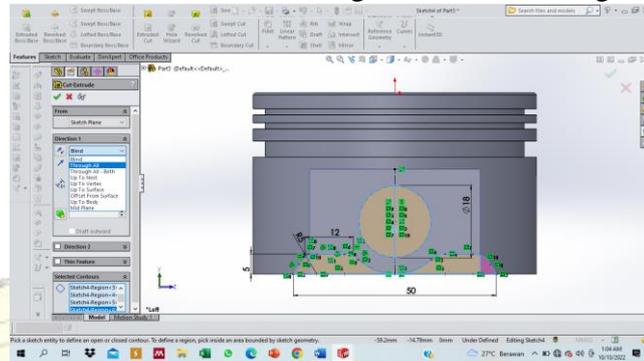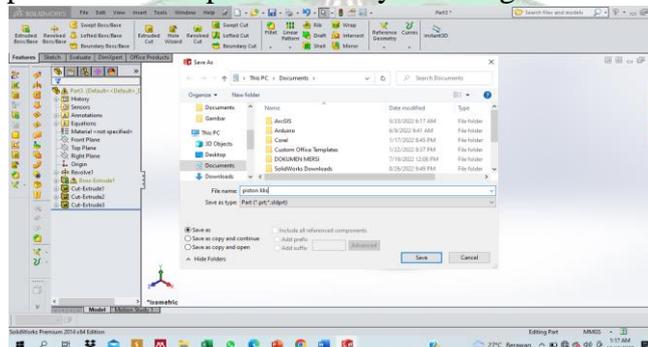

Picture 61. Piston head side view.

8. Then make a circle with the center point of the small circle that has been made before with a diameter of 20 mm then extruded cut then change the sketch plane on from to an offset of 1 mm and change the return direction to inward by clicking the arrow, then change the direction with a dept of 2 mm, then click ok. Then make it back on the opposite part in the same way.
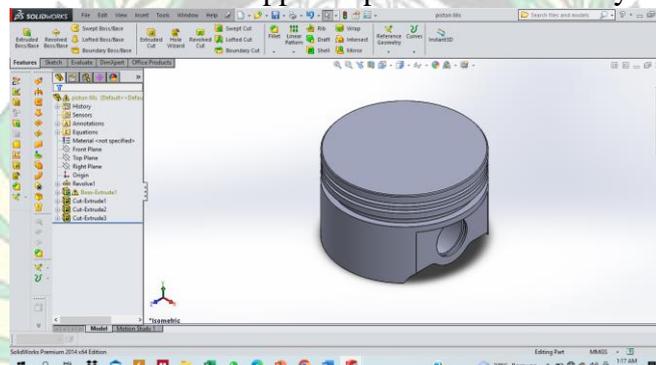
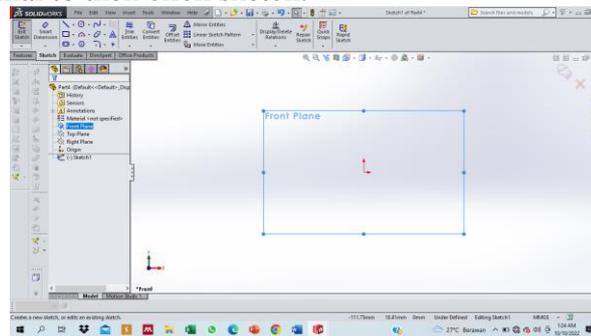Picture 62. Result display of hole making for piston pin.

9. The last step is to save the piston head by selecting the save command.


Picture 63. Piston head save view.

C. CONNECTING ROD

1. Select the file command then new, then select the part menu, then select front line then normal to then click sketch.


Picture 64. Front line view.

2. Make a connecting rod pattern by making a center line, then make a circle with the circle tool at each end of the line with a circle diameter of 18 mm right and 35 mm left and a line length of 130 mm, then select offset entities with a diameter of 4 mm, then make a line to form a connecting rod pattern using the line, fillet and trim entities tools to cut unused parts. Diameter as desired.
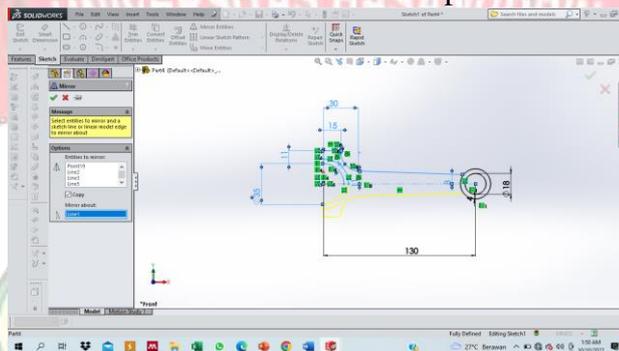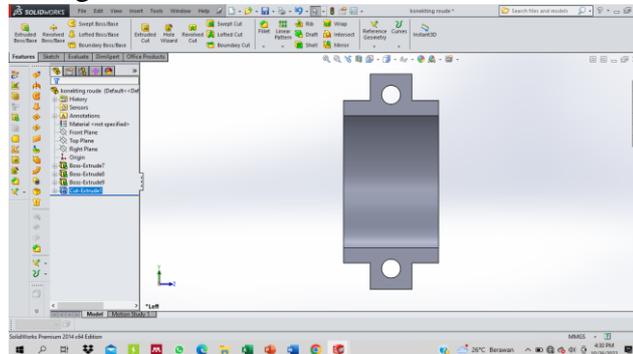

Picture 65. Sketch view of connecting rod.

3. The next step is to block all objects except the right circle, then select the mirror entities command, then click the center line, then click ok. Then select features, then select extruded boss/bass and do selected contours and click the connecting rod body and change the blind to mid plane with a diameter of 11 mm then click ok.
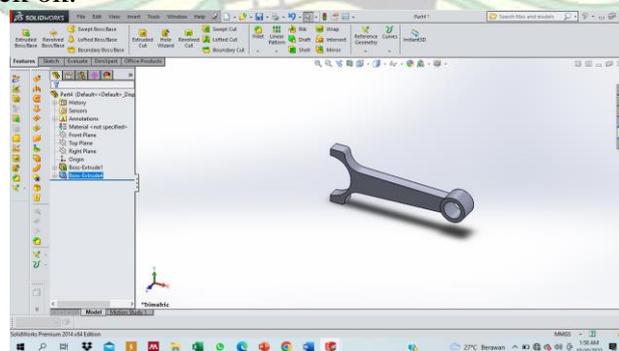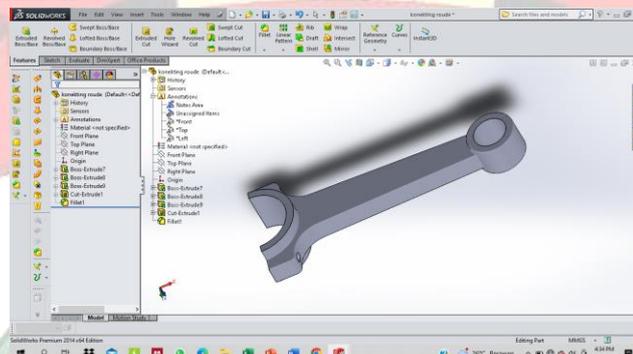

Picture 66. View of connecting rod before bore.

4. Next, make a hole on each side of the connecting rod end by making a line with the center line tool on the middle side of the top and bottom, then make a circle with a diameter of 5 mm, then select features then select extruded cut, change the blind to through all, then click ok.

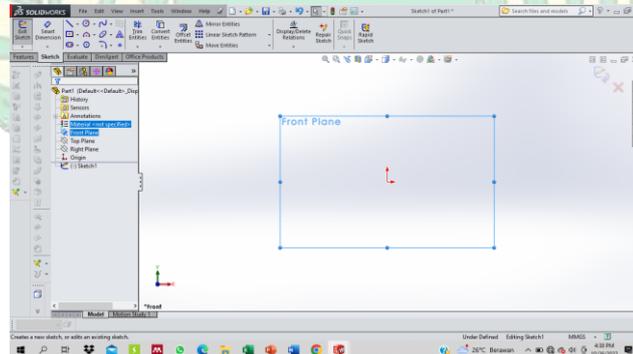Picture 67. Punching result display.

5. The last step is to save the results of making the connecting rod by selecting the save command.
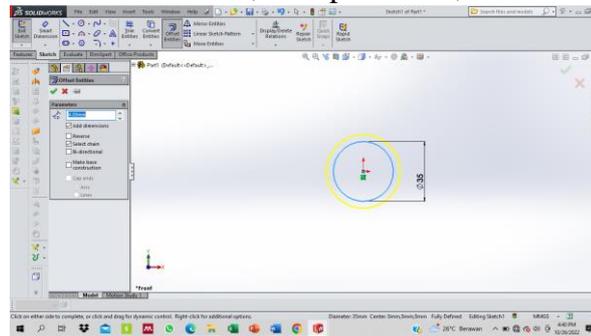

Picture 68. Final view of connecting rod.

D. ROD CAP
1. Select the file command then new, then select the part menu, then select front line then normal to then click sketch.


Picture 69. View of front line.

2. Create a circle with the circle tool with a diameter of 35 mm, then select offset entities with a diameter of 4 mm, then pull it out, then click ok.


Picture 70. Circle view.

3. Create a line with the line command as shown below starting from the left quadrant then to the right with a diameter of 11 mm long and 15 mm high, then trim entities at the top of the circle that are not used, then select extruded boss/bass then do selected contours and select the half circle, then remove the check mark on thin features, then change the blind to mid plane and diameter 25 mm, then click ok.
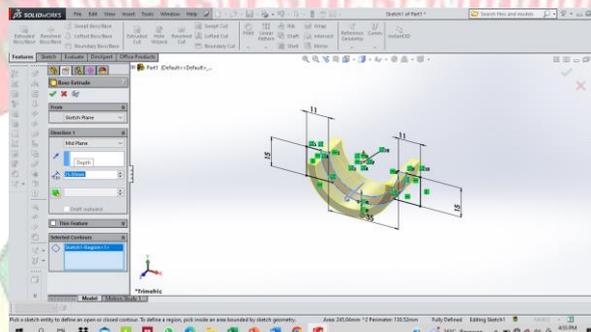

Picture 71. Result view of extruded boss/base.

4. The next step is to select the extruded boss/base command, then expand the dis entry, then select boss-extruded 1, then select sketch, then do selected contours, then select the left and right sides, then uncheck thin features, then change the blind to mid plane with a diameter of 11 mm, then ok.
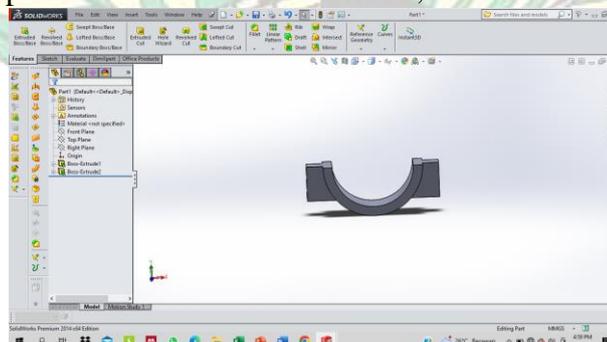

Picture 72. View of rod cap before hole.

5. Next, give holes on both sides of the rod cup by making auxiliary lines with the center line tool on the middle side of the top and bottom, then make a circle with a diameter of 5 mm, then select features then select extruded cut, change the blind to through all, then click ok.
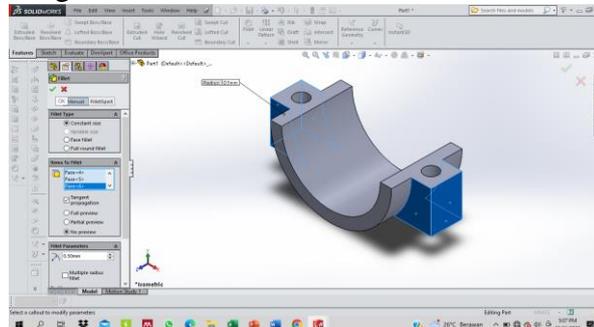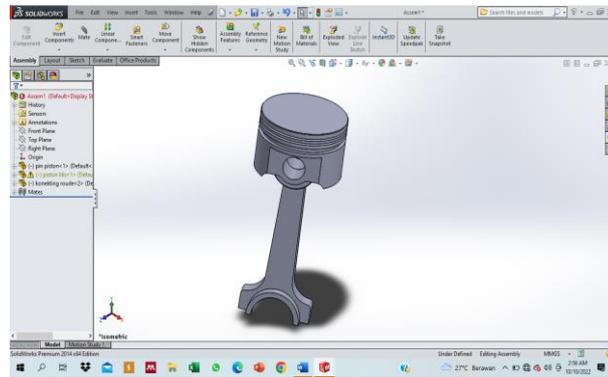

Picture 73. The appearance of the rod cup after being perforated.

6. The last step is to save the rod cup creation result by selecting the save command.
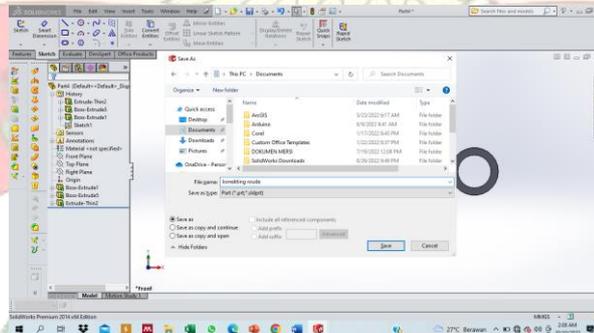
Picture 74. Display of saving rod cup.

Piston Making Results
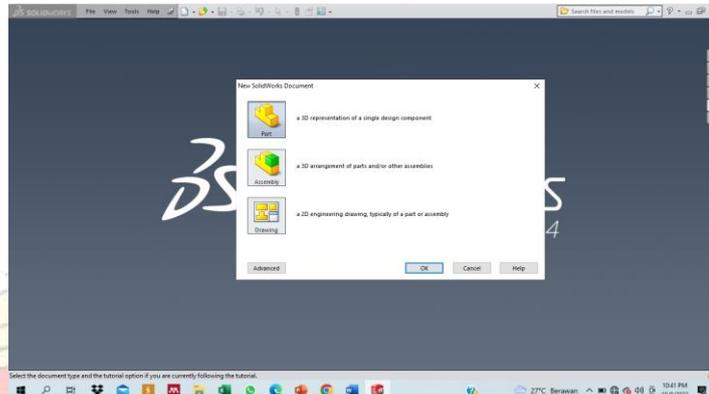

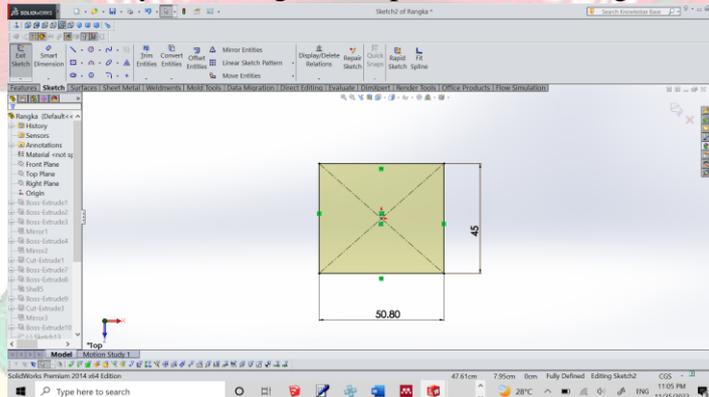Picture 75. Piston Making result display.

MINI PROJECT

Mini Project Making

1. Open SolidWorks software, press the new toolbar or press Ctrl+N. After the new SolidWorks document dialog box appears, select the part menu then ok.
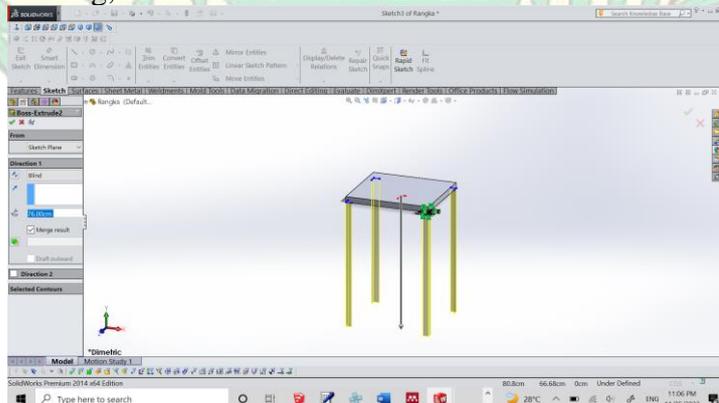

Gambar 76. SolidWorks initial window display.

2. Sketch the tool body according to the specified size using the line tools.
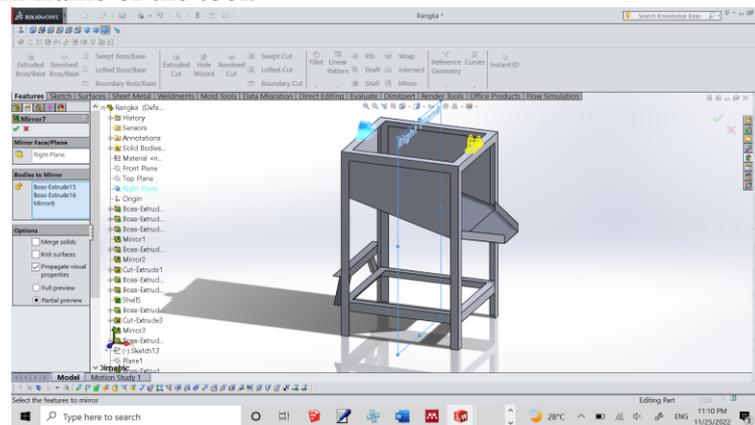

Gambar 77. Sketch view using the line tool.

3. Create a tool leg, then select features and then select extruded boss/bass.


Gambar 78. Tool leg creation view.

4. Create the side, inner frame and also support the foot of the tool using the line and mirror tools, then select features then select extruded boss/bass to display the main frame of the tool.
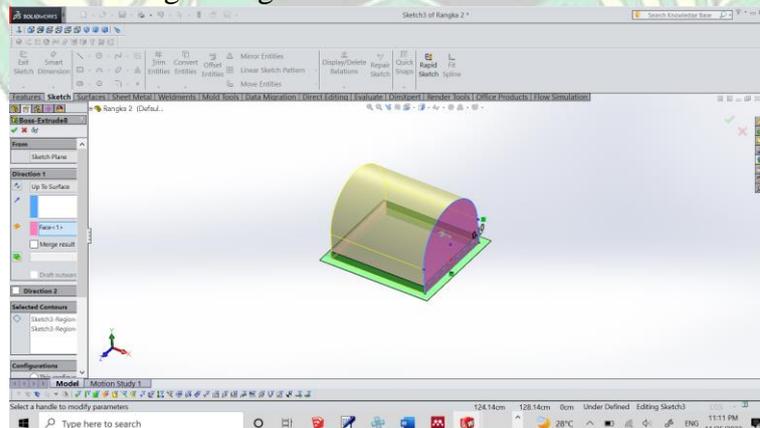

Gambar 79. Result view of extruded base/boss.

5. Create a tool cover design using line tools by giving dimensions according to the predetermined size.
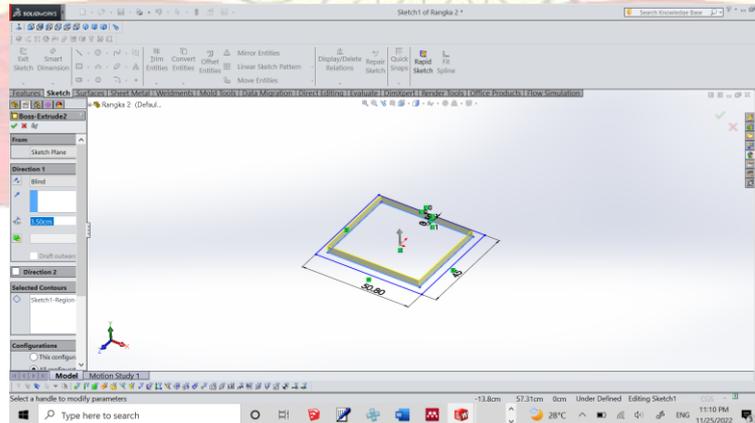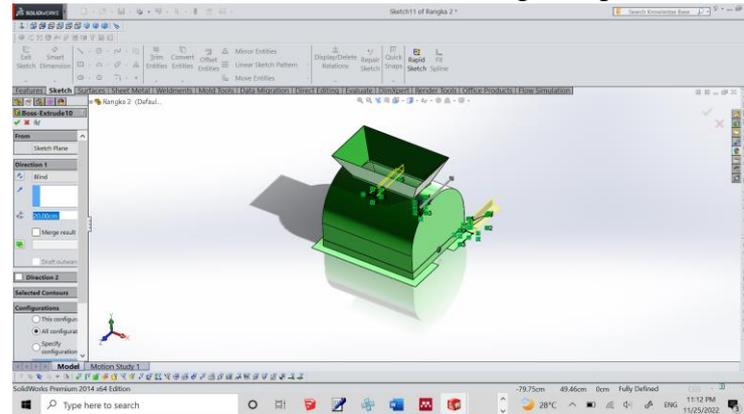

Gambar 80. Initial design view of the tool cover.

6. Create a cover design using the line tool and then select extruded boss/base.

Gambar 81. Tool cover display.

7. Create a hopper by selecting the line tool, then selecting extruded boss/bass, after that select the extruded cut command using the specified diameter.
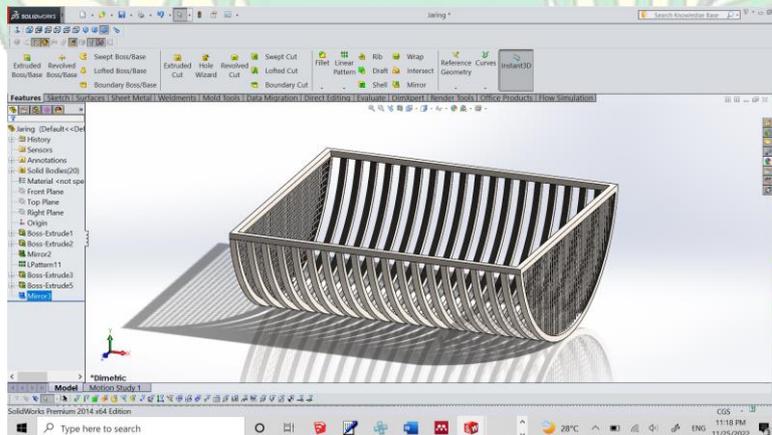

Gambar 82. Hopper circuit display.

8. Create a pulley circuit with a predetermined diameter using circle, then select the extruded boss/base command.
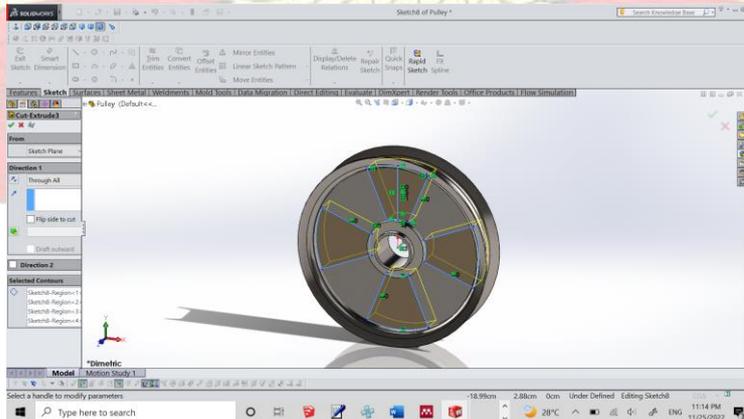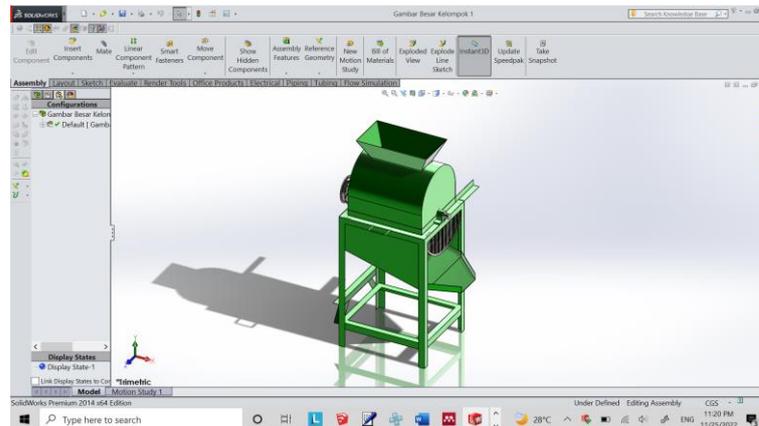

Gambar 83. Pulley design view.

9. Make the inside of the tool with a predetermined diameter using the line, circle, extruded boss/bass tools and the extruded cut command.

Gambar 84. Inside design view of the tool.

Mini Project Tool Making Results

Gambar 85. Display of mini project tool results.