

# Assessment of the Use of AutoCAD in Mechanical Engineering Technical Drawing Education

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**Abstract:** AutoCAD is one of the widely used software tools in engineering education. In this study, a general assessment of AutoCAD for the usage in the mechanical engineering technical drawing education is made. AutoCAD is assessed in terms of the fulfilment of the requirements defined for the main two technical drawing courses. AutoCAD is assessed in terms of its capability in meeting the requirements of the technical drawing courses.

**Keywords:** AutoCAD, assessment, technical drawing, education, mechanical engineering.

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## 1. Introduction

In general, there are mainly two technical drawing courses within the context of the mechanical engineering education. These courses are generally given using commercially available software, such as AutoCAD [1], Catia [2], DataCAD [3], DraftSight [4], Inventor [5], KeyCreator Direct CAD (former CADKEY) [6], NX (former Unigraphics) [7], OpenSCAD [8], Pro/Engineer [9], Solid Edge [10], Solid Works [11], Turbo-CAD [12] and VariCAD [13]. The first of the Computer Aided Drawing/Design (CAD) courses deals with basics of engineering drawing and focusses on part drawing (CAD I), while the second course (CAD II) captures assembly drawing and descriptive geometry. On the other hand, most of the Computer Aided Engineering (CAE) software used in engineering education requires knowledge of solid modelling, which is covered by CAD I and CAD II. It is common to use a CAD software for these purposes, and AutoCAD [1] is one of the widely used software tools within this context.

In this study, the AutoCAD Mechanical (2013) software – a version of AutoCAD with some features specialized in mechanical engineering - is assessed for the use in mechanical engineering technical drawing education. This study is not a benchmarking for comparing CAD software tools in mechanical engineering education, but focuses on the assessment of the usage of AutoCAD, which is considered as the most widely used software for engineering education.

## 2. Methodology

In this study, a three-step approach has been used.

Step 1 – Define contents of the CAD courses,

Step 2 – Determine requirements for each content,

Step 3 – Assess the capability of AutoCAD Mechanical in implementing each requirement.

### 2.1. Step 1 – Defining Contents of the CAD Courses

In defining the CAD I and CAD II course contents, contents of technical drawing courses ME113 Computer Aided Engineering Drawing I and ME114 Computer Aided Engineering Drawing II that are given to mechanical engineering students of Çankaya University have been used as a basis. The contents are given in syllabi of the courses [14,15] and summarized in TABLE 1.

The CAD I and CAD II courses of Cankaya University were established in 2011, after a comprehensive survey of many international and national universities' CAD courses. A comparison of the course contents of some of the selected international and national universities is summarized in

Table 2 and TABLE 3 for the CAD I and CAD II courses, respectively. Considering the comparison tables, the contents could be regarded as a basis for the assessment study and as samples representing the general CAD I and CAD II course contents, rather than belonging to the courses of a single university/department.

TABLE 1. Contents of CAD I and CAD II Courses

Number	CAD I	CAD II
1	Basics of engineering drawing	Introduction to solid modelling
2	Coordinate systems	Construction of solid features
3	Basic geometric elements	Tolerances, surface quality marks

4	Drawing simple geometric objects and editing	Geometric dimensioning and tolerancing
5	Solid modelling fundamentals	Working drawing and assemblies
6	Projection theory/multi-views from 3-D	Threaded fasteners
7	Projection of 3 <sup>rd</sup> view from others	Non-threaded fasteners, springs
8	Pictorial projection	Mechanisms
9	Section view	Welding
10	Dimensioning and tolerancing	Descriptive geometry
11	Basics of assembly drawing	
12	Presentation of engineering drawings/printing	

TABLE 2. A Comparison of the CAD I Course Contents of Some Universities.

Number	CAD I Course Content	Course Content of the University's Mechanical Engineering Department									
		1*	2	3	4	5	6	7	8	9	10
1	Basics of engineering drawing	+	+	+	+	+	+	+	+	+	+
2	Coordinate systems	+	+	+	+	+	+	+	+	+	+
3	Basic geometric elements	+	+	+	+	+	+	+	+	+	+
4	Drawing simple geometric objects and editing	+	+	+	+	+	+	+	+	+	+
5	Solid modelling fundamentals	+	-	-	+	+	-	-	-	+	-
6	Projection theory/multi-views from 3-D	+	+	+	+	+	+	+	+	+	+
7	Projection of 3 <sup>rd</sup> view from others	+	+	+	+	+	+	+	+	+	+
8	Pictorial projection	+	+	+	+	+	+	+	+	+	+
9	Section view	+	+	+	+	+	+	+	+	+	+
10	Dimensioning and tolerancing	+	+	+	+	+	+	+	+	+	+
11	Basics of assembly drawing	+	+	+	+	-	+	-	-	+	-
12	Presentation of engineering drawings/printing	+	+	+	+	+	+	+	+	+	+

\*Numbers represent: 1- Cankaya, 2-UC, Berkeley, 3-Cambridge, 4-Carnegie Melon, 5-Gazi, 6-ITU, 7-IYTE, 8-METU, 9-Stanford, 10-YTU universities.

TABLE 3. A Comparison of the CAD II Course Contents of Some Universities.

Number	CAD II Course Content	Course Content of the University's Mechanical Engineering Department							
		1*	2	3	4	5	6	7	8
1	Introduction to solid modelling	+	+	+	+	+	+	+	+
2	Construction of solid features	+	+	+	+	+	+	+	+
3	Tolerances, surface quality marks	+	+	+	+	+	+	+	+
4	Geometric dimensioning and tolerancing	+	+	+	+	+	+	+	+
5	Working drawing and assemblies	+	+	+	+	+	+	+	+
6	Threaded fasteners	+	+	+	+	+	+	+	+
7	Non-threaded fasteners, springs	+	+	+	+	+	+	+	+
8	Mechanisms	+	+	+	+	+	+	+	+
9	Welding	+	+	+	+	+	+	+	+
10	Descriptive geometry	+	-	-	+	-	-	+	-

\*Numbers represent: 1- Cankaya, 2-UC, Berkeley, 3-Carnegie Melon, 4-Gazi, 5-ITU, 6-IYTE, 7-METU, 8-YTU universities.

The contents of the courses are given at least in two levels. TABLE 1 shows only the first level of contents. An example of the second level is given in TABLE 4 for 'Drawing Simple Geometric Objects and Editing'.

TABLE 4. Second Level of Contents for Simple Geometric Objects and Editing

Number	Content
4.1	Drawing a rectangle
4.2	Drawing a circle
4.3	Drawing tangent lines or arcs
4.4	Drawing an arc
4.5	Drawing an ellipse
4.6	Drawing an involute
4.7	Drawing regular polygons
4.8	Creating a copy of an object
4.9	Creating a mirror copy of an object
4.10	Creating a resized copy of an object at a distance
4.11	Trimming unnecessary portions of a drawing
4.12	Extending incomplete lines
4.13	Changing size of an object
4.14	Chamfering or filleting
4.15	Creating array of an object
4.16	Moving and rotating an object
4.17	Undoing, re-doing, erasing, zooming

## 2.2. Step 2 – Determining Requirements for Contents of the CAD Courses

For each content, the requirements that mechanical engineering students should achieve are determined. As an example, requirements for the content ‘simple geometric objects and editing’ in CAD I is given in TABLE 5.

TABLE 5. Requirements for Simple Geometric Objects and Editing

Number	Requirement
R4.1	Draw a rectangle
R4.2	Draw a circle
R4.3	Draw an arc
R4.4	Draw lines or arcs which are tangent to two circles or lines
R4.5	Draw an ellipse
R4.6	Draw an involute
R4.7	Draw equilateral polygons
R4.8	Create a single or multiple copy of an object
R4.9	Create a mirror copy of an object
R4.10	Create a resized copy of an object at a defined distance
R4.11	Trim unnecessary portions of a drawing
R4.12	Extend incomplete lines until a boundary item
R4.13	Change size of an object
R4.14	Chamfer or fillet the sharp edges of an object
R4.15	Create array of an object in rectangular or polar form
R4.16	Move an object and rotate at an angle
R4.17	Undo, re-do, erase, zoom

There are 104 and 73 requirements for CAD I and CAD II, respectively.

## 2.3. Step 3 – Assessing Capability of AutoCAD Mechanical in Implementing Each Requirement

AutoCAD Mechanical is assessed on a ‘1 or 0’ basis by reviewing whether it does or does not meet each requirement. If the assessment is partial ‘1’ and partial ‘0’, the requirement is fragmented so that the result is either ‘1’ or ‘0’.

For example, in assessing AutoCAD Mechanical for the requirements of unit system, the requirement ‘Metric, US Customary or any other unit for length and angle can be defined or be changed at any time’ is rated as 1, which indicates that AutoCAD Mechanical meets the requirement. The assessment is not the subjective view of the assessor. It shows only whether students can or cannot implement the requirement by using AutoCAD Mechanical.

The assessment of AutoCAD Mechanical is done for the CAD I and CAD II courses separately.

### **3. Results and Discussion**

#### **3.1. Assessment of AutoCAD Mechanical in Terms of Fulfillment of the Requirements of a CAD I course**

The main subjects of the course, the requirements, and assessment of AutoCAD Mechanical in meeting the requirements are given below. Numbered items define the main subjects of the course, while R and A stand for the requirements in that area and assessment of AutoCAD Mechanical in fulfilling the requirement, respectively. The score '1 or 0' is given at the end of each assessment. The score '1 and 0' imply that AutoCAD Mechanical does or does not fulfil the requirement.

##### **3.1.1. Basics of Engineering Drawing**

R1.1. Determine units for drawing

A1.1. Metric, US Customary or any other unit for length and angle can be defined or be changed at any time. In AutoCAD, there are three ways to implement a drawing action, namely writing a command, selecting a menu item or clicking on the related toolbar icon, and then following instructions of the program (1).

R1.2. Establish a layout page

A1.2. Layout page is required to take a printout of the drawing. For each drawing in the model space, many layout pages with different paper sizes, units, scales and title blocks can be created using the templates of AutoCAD or established by the user.

R1.2.1. Determine paper size

A1.2.1. The paper size may change with the units used, complexity of the drawing, and the standards adopted. Templates can be prepared for different purposes and paper sizes, and then a convenient template can be selected for a new layout (1).

R1.2.2. Determine drawing scale

A1.2.2. The scale used in the model space is 1:1. A different scale may be used to fit the drawing in the layout page. The scale will change with the page size (1).

R1.2.3. Develop title block

A1.2.3. AutoCAD has some title block templates. These templates are not proper for professional use. However, users can create their own title block templates which will be in line with ISO, ANSI or any international standard (1).

### R1.3 Prepare a template

A1.3. AutoCAD has many templates for different uses. Any available template can also be used or new templates can be created with a previously defined setup. All the parameters such as unit, limits of drawing area, grid size, snap spacing, layout templates can be defined in the templates (1).

## **3.1.2. Coordinate Systems**

### R2.1. Define a coordinate system

A2.1. A cartesian or polar coordinate system can be used in the world coordinate system or user (local) coordinate system. Depending on the information available, absolute or relative coordinate values can be used. Relative coordinates are very practical, and can be used with either Cartesian or Polar coordinates.

#### R2.1.1. Locate a point by using Cartesian absolute coordinate values

A2.1.1. A point can be located by inputting absolute Cartesian coordinate values in the world or local coordinate system (1).

#### R2.1.2. Locate a point by using Cartesian relative coordinate values

A2.1.2. A point can be located by inputting relative Cartesian coordinate values in the world or local coordinate system (1).

#### R2.1.3. Locate a point by using polar absolute coordinate values

A2.1.3. A point can be located by inputting absolute polar coordinate values in the world or local coordinate system (1).

#### R2.1.4. Locate a point by using polar relative coordinate values

A2.1.4. A point can be located by inputting relative polar coordinate values in the world or local coordinate system (1).

## **3.1.3. Basic Geometric Elements**

### R3.1. Draw a point

A3.1. Point command, clicking on point icon or selecting the point menu item requires coordinates of the point, inputting the coordinates through writing coordinate values or clicking on the desired location (1).

### R3.2. Create a grid system

A3.2. The grid system of pattern of dots is easy to create. It helps the user in aligning objects and measuring the distances between the grid dots. Grid spacing can be changed. The grid system can be turned on or off. Grid snapping is a very useful tool in drawing a defined sized element. Snapping restricts the motion of the cross hair to the defined grid spacing intervals. The orientation of the grid axes can be rotated (1).

### R3.3. Draw a line

A3.3. A line can be drawn easily using different ways. It may be a finite line, ray or infinite line. Line conditions such as being parallel, perpendicular, tangent or intersecting is easy to define. Basic types of lines such as visible line, hidden line, center line, dimension line, extension line, cutting-plane line, phantom line can be drawn easily with AutoCAD, especially with AutoCAD Mechanical. Basic types of lines are shown in FIGURE 1.

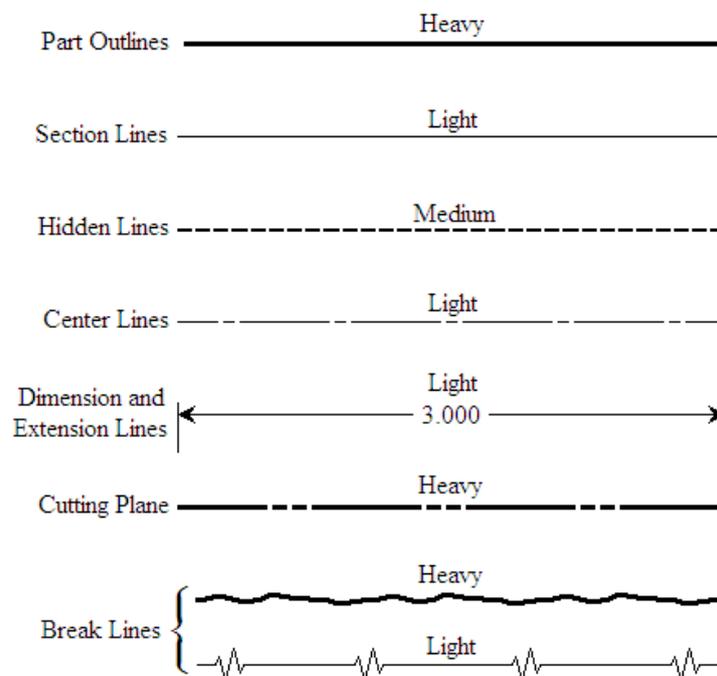


FIGURE 1. Basic line types.

#### R3.3.1. Draw a line by using Cartesian coordinate system

A3.3.1. Using a relative coordinate system is easier and more practical than an absolute coordinate system. After inputting the starting point coordinate values, the line can be drawn

by inputting  $\Delta x$ ,  $\Delta y$ ,  $\Delta z$  values of the next point with respect to the previous point (1).

### R3.3.2. Draw a line by using the polar coordinate system

A3.3.2. The polar relative coordinate system is default in AutoCAD Mechanical. After clicking on the first point location, the line is drawn by inputting distance and angle of the second point with respect to the first (1).

### R3.4. Arrange precedence of lines

A3.4. Lines which are at the same location can be arranged in the order as visible line, hidden line and center line (1).

### R3.5. Draw polylines in different forms

A3.5. The polyline tool is a very flexible and useful tool combining different portions of lines as a single object, which can be split later. Changing width and shape of the line is possible as shown in FIGURE 2. Line types can be changed with the 'pedit' command. There is a mix of some line definitions: AutoCAD has two options for spline, namely 'spline-fit points' which is identical to mathematical spline, and 'spline-control vertices' which is similar to a Bezier curve.

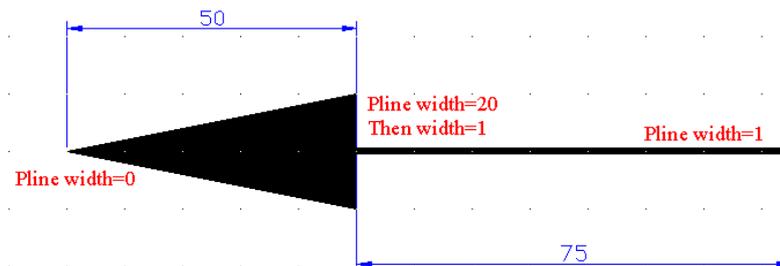


FIGURE 2. Polyline example.

#### R3.5.1. Draw a line as a polyline

A3.5.1. It is the same as drawing a line. The difference is that all the lines in polyline are treated as a single object, and there is no need to use the 'region' command to create a surface before creating a 3-D model (1).

#### R3.5.2. Draw an arc as a polyline

A3.5.2. It is the same as drawing an arc. In polyline, arc is a part of polyline and treated as a single object (1).

### 3.1.4. Drawing Simple Geometric Objects and Editing

R4.1. Draw a rectangle

A4.1. Rectangles can be drawn with various line widths, with normal, chamfered or filleted corners as shown in FIGURE 3 (1).

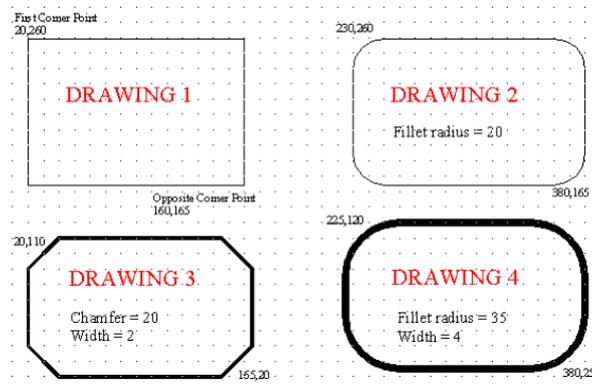


FIGURE 3. Rectangle types.

R4.2. Draw a circle

A4.2. Drawing a circle by using different ways depending on the available data is possible. It might be circumscribed about or inscribed in a polygon. It might be concentric or eccentric. AutoCAD provides six options to draw a circle: center point-radius, center point-diameter, 2 point, 3 point, tangent-tangent-radius, and tangent-tangent-tangent. An example of tangent-tangent-radius circle application is given in FIGURE 4 (1).

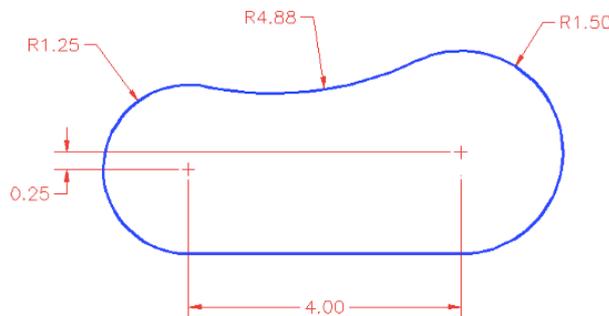


FIGURE 4. Tangent-tangent-radius circle application.

R4.3. Draw an arc

A4.3. Arcs can be drawn using different combinations of center, end point, start point, radius, angle, length, and direction data (1).

R4.4. Draw lines or arcs which are tangent to two circles or lines

A4.4. AutoCAD provides convenient object snap for finding tangency so that drawing a line tangent to two circles, an arc tangent to one or two lines or arcs is a simple process.

R4.4.1. Draw a line which is tangent to an arc or circle

A4.4.1. A tangent line is drawn by inputting the first point location, then going over the circle or arc, and clicking at the location where the tangency sign appears (1).

R4.4.2. Draw an arc or circle which is tangent to the other two circles or arcs

A4.4.2. It can be drawn by for example drawing a circle using tangent-tangent-radius option (1).

R4.5. Draw an ellipse

A4.5. There are three options in drawing an ellipse, namely axis end point, center, and arc. An example with center option is shown in FIGURE 5 (1).

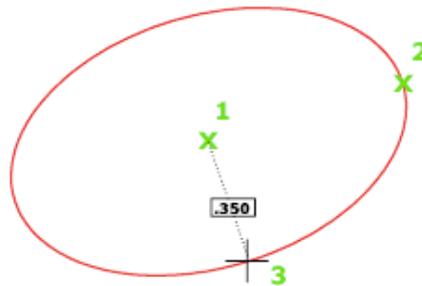


FIGURE 5. An ellipse with center option.

R4.6. Draw an involute

A4.6. Involute – the path of a point on a string as the string unwinds from a line, polygon, or circle as shown in FIGURE 6- can be drawn using tangencies and snapping (1).

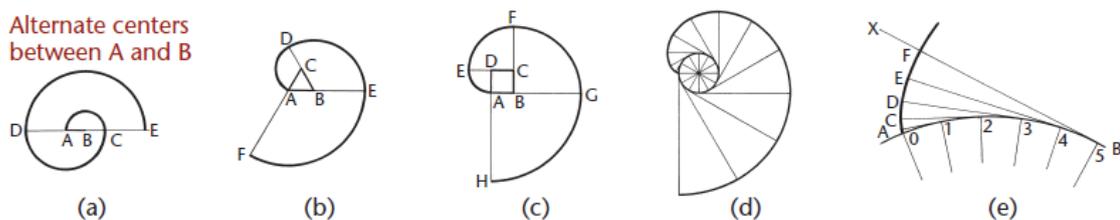


FIGURE 6. Involute examples.

R4.7. Draw equilateral polygons

A4.7. Regular polygons such as the ones given in FIGURE 7 can easily be drawn using the options of center or edge, and inscribed in or circumscribed about a circle (1).

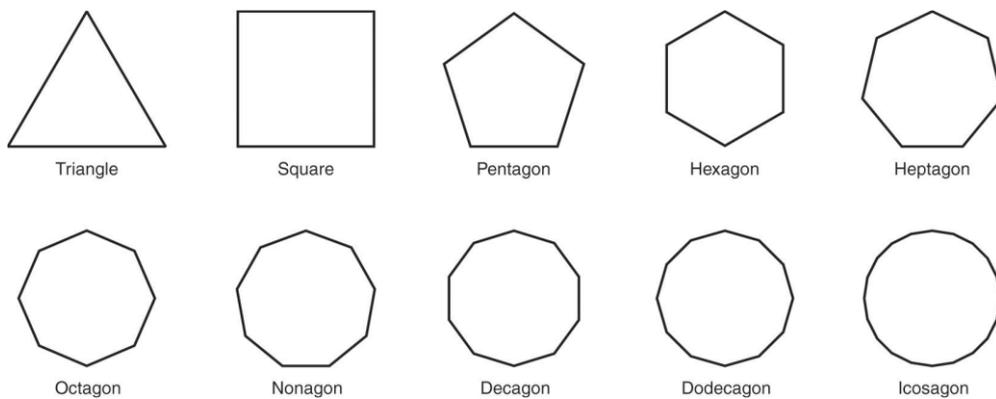


FIGURE 7. Regular polygons.

R4.8. Create a single or multiple copy of an object

A4.8. Objects can be copied as many as required at the desired locations (1).

R4.9. Create a mirror copy of an object

A4.9. An object having symmetry can be created through mirror copying of its quarter or half as shown in FIGURE 8 (1).

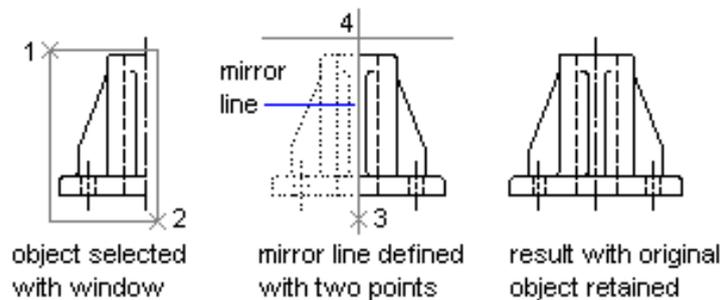


FIGURE 8. Mirror copying example.

R4.10. Create a resized copy of an object at a defined distance

A4.10. AutoCAD creates the resized copies at the defined distance easily using ‘offset’ command as shown in FIGURE 9 (1).

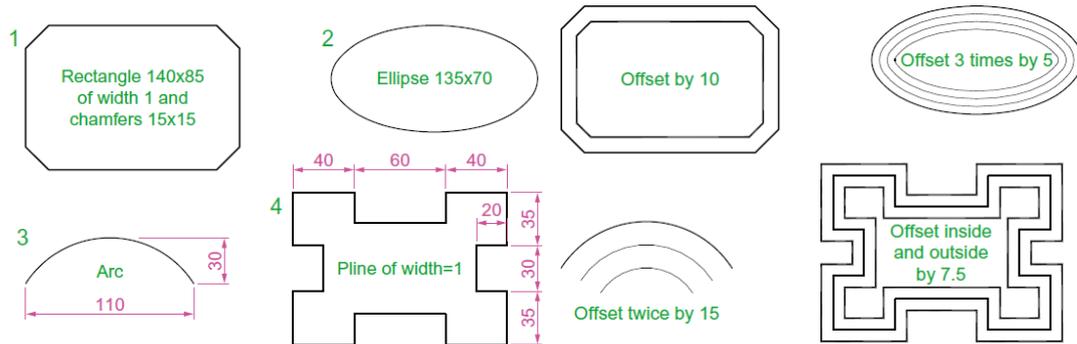


FIGURE 9. Offset examples.

R4.11. Trim unnecessary portions of a drawing

A4.11. Trimming and deleting unrequired portions of a drawing can easily be done as shown in FIGURE 10 (1).

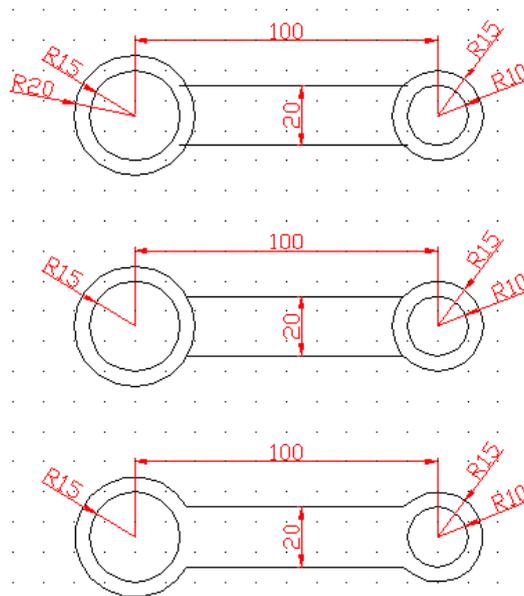


FIGURE 10. An example of trimming.

R4.12. Extend incomplete lines until a boundary item

A4.12. Lines can be extended up to a defined object feature using 'extend' command as shown in FIGURE 11 (1).

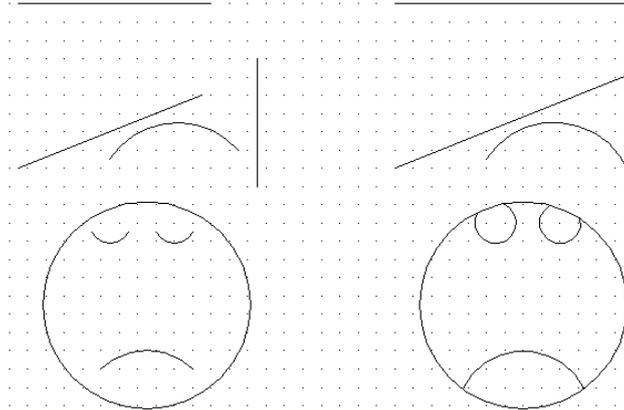


FIGURE 11. An example of extend.

#### R4.13. Change size of an object

A4.13. There are many ways to change the size, such as changing the value in the properties table, using 'lengthen' or 'stretch' command, or changing scale with a scale factor (1).

#### R4.14. Chamfer or fillet the sharp edges of an object

A4.14. Sharp corners can be truncated or rounded.

##### R4.14.1. Chamfer the sharp vertices

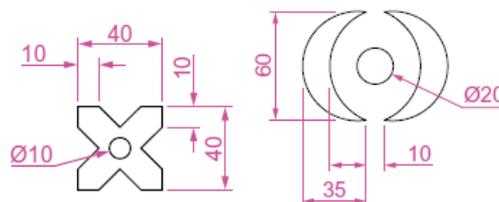
A4.14.1. Chamfering can be done using 'two distances' or 'a distance and angle' options (1).

##### R4.14.2. Round the sharp vertices

A4.14.2. By using 'fillet' command and defining a radius, a sharp edge can be rounded easily (1).

#### R4.15. Create array of an object in rectangular or polar form or along a path

A4.15. Multiple copies of an object can be created using rectangular or polar array commands as shown in FIGURE 12. It is also possible to multiply an object along a curved path. Basic features can be arranged and be changed at any time.



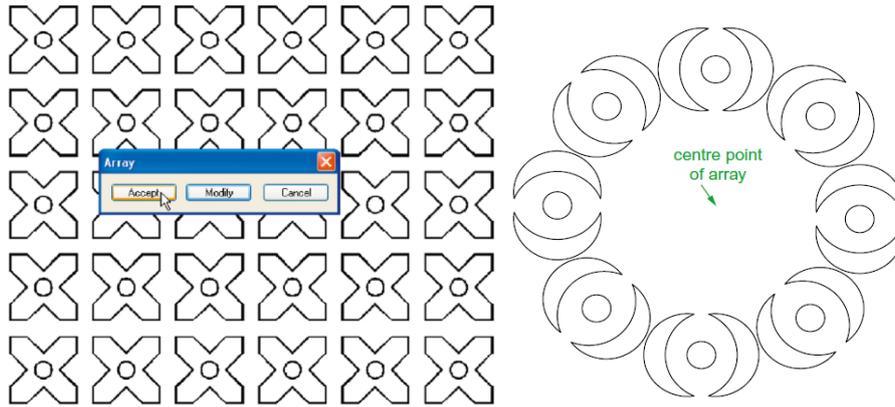


FIGURE 12. Example of rectangular and polar arrays.

R4.15.1. Create an array of an object in rectangular form

A4.15.1. After creating a base view to be multiplied, it can be arranged in linear array form by defining the number of rows, distance between rows, number of columns and the distance between the columns (1).

R4.15.2. Create an array of an object in polar form

A4.15.2. After creating a base view to be multiplied, it can be arranged in polar array form by defining the center of array, number of items and angle of rotation (1).

R4.15.3. Create an array of an object about a curved path

A4.15.3. After creating a base view to be multiplied, it can be multiplied about a curved path. Number of objects and distances between the array elements can be defined (1).

R4.16. Move an object and rotate at an angle

A4.16. Objects can be moved to a desired location and be rotated about a point at a defined angle in counter clockwise or clockwise direction like the one given in FIGURE 13 (1).

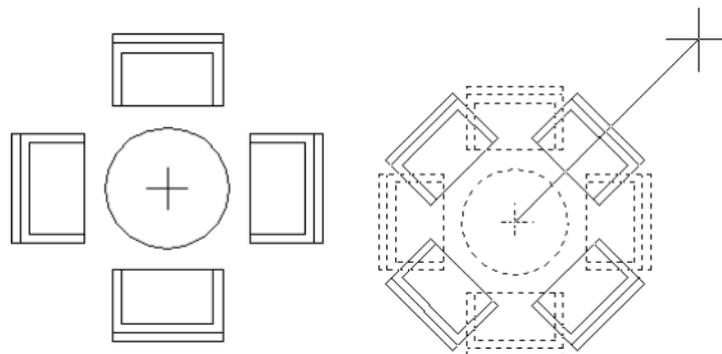


FIGURE 13. Rotating an object at an angle.

R4.17. Undo, re-do, erase, zoom

A4.17. By using the basic handling tools, any number of actions can be taken back or redone, view of the drawing can be zoomed in or out. Any portion of the drawing can easily be deleted (1).

### 3.1.5. Solid Modeling Fundamentals

R5.1. Create primitive solid models

A5.1. Primitive solids such as box, cone, cylinder, torus, wedge, pyramid, and sphere can be created easily by inputting the required data (1).

R5.2. Create a complex solid by using extrusion technique.

A5.2. Extrude tool can be used along a complex path, or perpendicular/inclined line at an angle path. An example is given in FIGURE 14 (1).

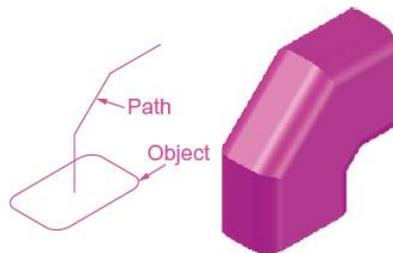


FIGURE 14. Use of extrude tool.

R5.3. Create a complex solid by using revolving technique

A5.3. Revolve tool can be used in creating objects with axial symmetry. An example drawing is shown in FIGURE 15 (1).

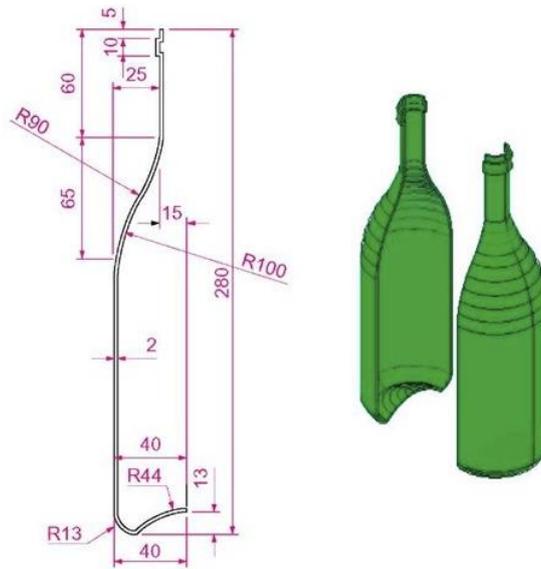


FIGURE 15. Use of revolve tool [16].

R5.4. Create a complex solid by using sweeping technique

A5.4. The sweep tool can be used to create complex solid models. For example, sweeping a defined profile along a helix may create a spring model or teeth model of a bolt or screw. An example is shown in FIGURE 16 (1).

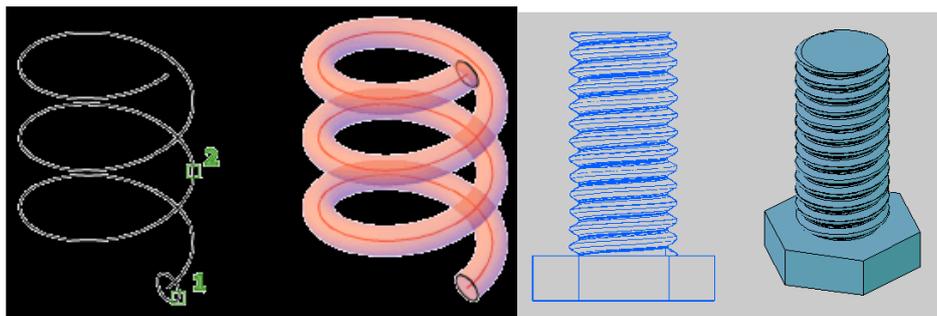


FIGURE 16. Use of sweep tool.

R5.5. Create a complex solid by using lofting technique

A5.5. Loft tool can be used to draw very complex solid models. An example is given in FIGURE 17 (1).

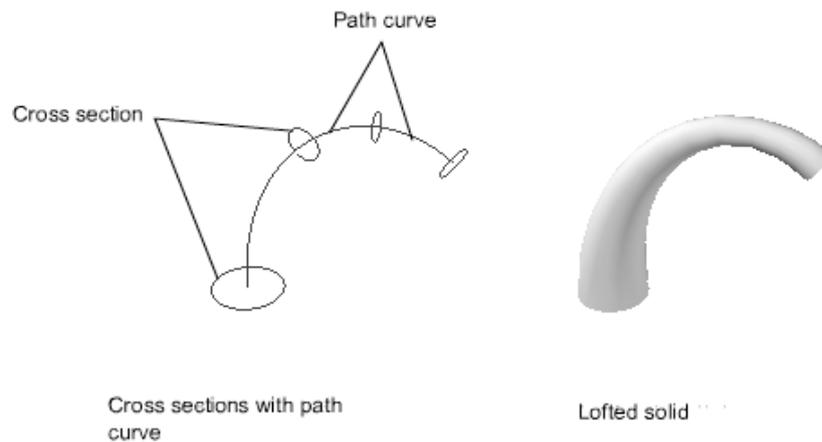


FIGURE 17. Use of loft tool.

R5.6. Create a solid model by Constructive Geometry Solid Modelling

A5.6. Solid models of complex geometric objects can be created using basic geometric forms and Boolean operators. An example showing the steps of creating the object is given in FIGURE 18 (1).

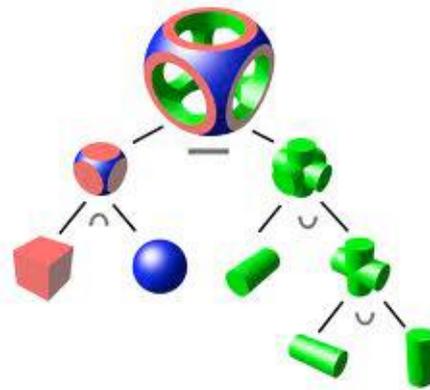


FIGURE 18. Constructive Geometry Solid Modelling Example.

R5.7. Create solid model of an object using Boolean operators

A5.7. Boolean operators, namely union, subtraction, and intersection, can be effectively used in creating new complex objects from comparatively simple objects like the one shown in FIGURE 19. Exclusive-or (XOR) option is not available in AutoCAD.

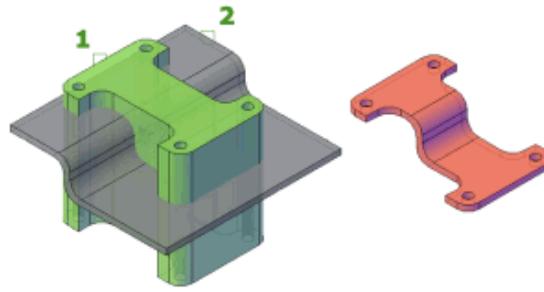


FIGURE 19. Use of Boolean operators [17].

#### R5.7.1. Create solid model by using Boolean ‘union’ operation

A5.7.1. A new complex model can be created from two separate parts by adding the parts, using union command (1).

#### R5.7.2. Create solid model by using Boolean ‘subtraction’ operation

A5.7.2. A new complex model can be created from two separate parts by subtracting one from the other, using subtract command (1).

#### R5.7.3. Create solid model by using Boolean ‘intersection’ operation

A5.7.3. A new complex model can be created from two separate parts by keeping intersecting portions, using intersect command (1).

#### R5.7.4. Create solid model by using Boolean ‘Exclusive or (XOR)’ operation

A5.7.4. XOR is a combination of Boolean operations ‘union’ and intersection. In the intersection operation common portions of the two parts are reserved. In XOR operation, the deleted portion of the two parts after application of intersection operation is reserved. It is the union minus the intersection of two parts. That command is not available with AutoCAD (0).

#### R5.8. Create Constraint Based solid models

A5.8. Constraint based (parametric) modelling in 2-D can be used effectively. Changing the parameters changes the drawing automatically while keeping the constraints. Two examples are given in FIGURE 20. Parametric modelling in 3-D is not possible.

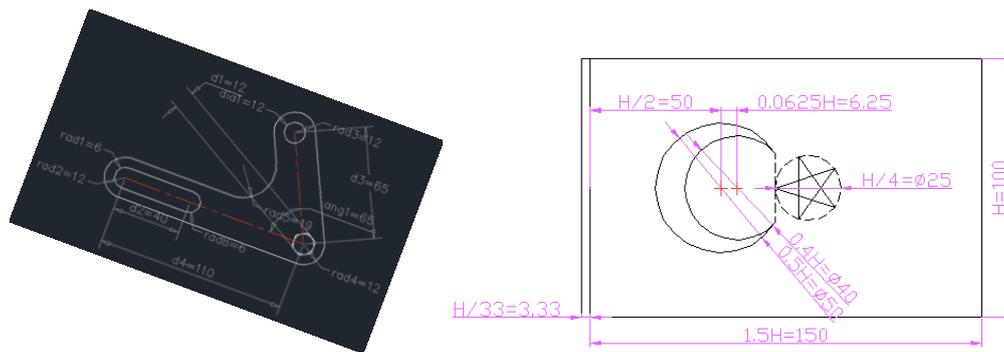


FIGURE 20. Constraint based modelling example for 2-D.

#### R5.8.1. Create solid models by using parametric 2-D models

A5.8.1. AutoCAD is very flexible in defining parametric constraints on any plane. After changing the parameters, new sketch file to be used in creating solid model is generated automatically without a need to redraw the 2-D profile (1).

#### R5.8.2. Create solid models by using parametric 2-D models

A5.8.2. AutoCAD is not intended for 3-D modelling, and Autodesk, who is the owner of AutoCAD provides the software Inventor for this purpose. AutoCAD does not keep the sketch which is used to create a 3-D model. When the 3-D model is created, the sketch is converted to 3-D model and no sketch is contained in the 3-D model data, but it is kept in the 3-D model data by the solid modelling software such as Inventor of Autodesk. For example, the 3-D parametric design shown in FIGURE 21 is not possible with AutoCAD, since AutoCAD provides only 2-D parametric capability. Any feature in the 3<sup>rd</sup> dimension cannot be defined as a parameter of the features created in the 2-D sketch plane. After generating a 3-D model, the defined parameters are cancelled and the parametric model is converted to a 3-D model which has no parametric data. A standard part or assembly in different sizes is easy with 3-D parametric design, which can be created by changing only the values of the parameters. In AutoCAD a different size of a product should be re-drawn. This feature is more important in mating parts of an assembly, since any configuration change in one part requires related changes in the other parts. This can be achieved easily through 3-D parametric modelling. Absence of this feature is a deficiency of AutoCAD (0).

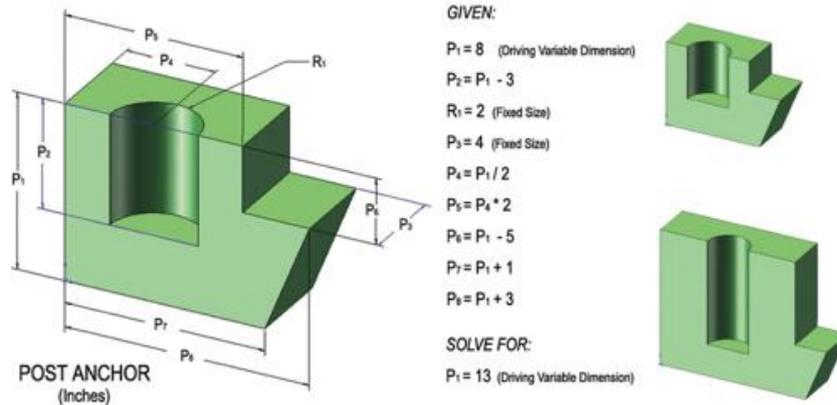


FIGURE 21: Parametric modeling in 3-D [18].

R5.9. Create 3-D models by using sweeping techniques

A5.9. Sweeping a plane (generatrix) along a path (diretrix) creates complex solid geometries, and using Boolean operations may yield to the desired very complex solid model. An example is given in FIGURE 22. Before creating the solid model, the generatrix should be drawn on the proper working plane. Working plane can be defined using different techniques such as using user coordinate system or object snapping utility. Setting UCSFOLLOW to 1 provides working in 2-D instead of 3-D space (1).

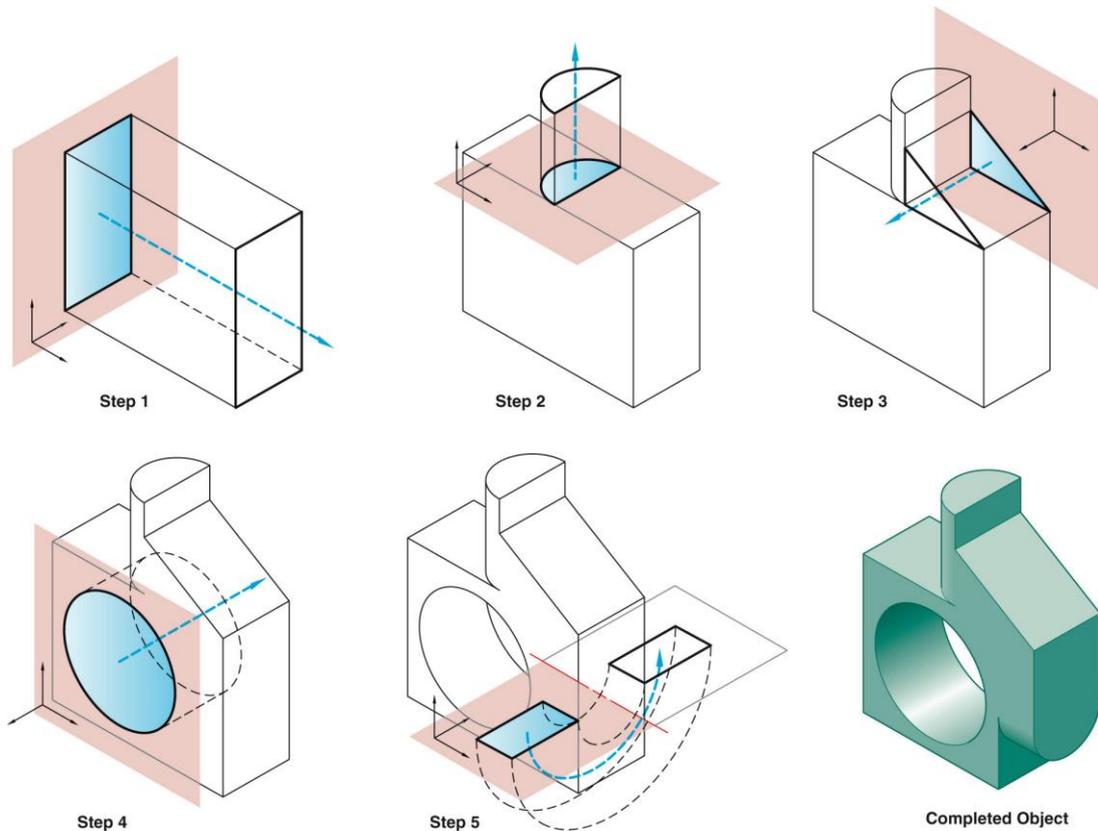


FIGURE 22. Steps of creating a solid model using sweeping and Boolean operations [19].

R5.10. Duplicate solid part features.

A5.10. Solid parts can be duplicated using linear and radial arrays like the one given in FIGURE 23. Creating array of holes requires subtracting (1).

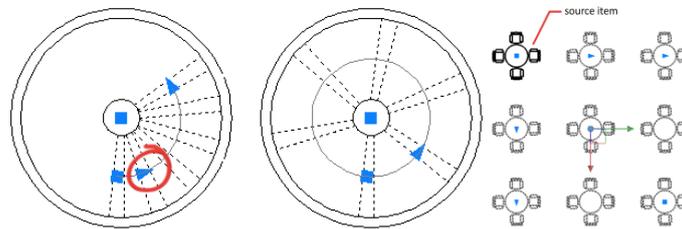


FIGURE 23. Linear and polar array application [20].

R5.11. Obtain volume related properties from the drawing

A5.11. Volume related properties such as volume, mass, moments of inertia, and centroid cannot be obtained from the solid model (0).

R5.12. Use different options for rendering a model

A5.12. AutoCAD has many options in rendering a model, such as 2D wireframe, conceptual, hidden, realistic, shaded, shaded with edges, shades of grades, sketchy, wireframe, and x-ray. Some rendering options are shown in FIGURE 24 (1).

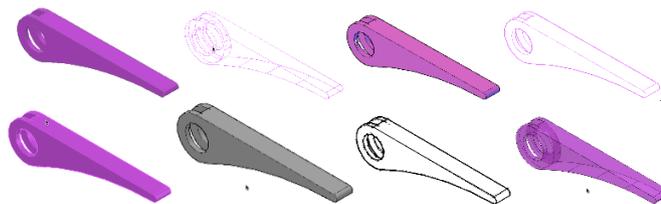


FIGURE 24. Rendering options.

R5.13. Get the solid models of standard parts from the content library

A5.13. AutoCAD is not intended to create solid models although it has basic tools for creating solid geometries. AutoCAD has 2-D multi-views of standard parts. However, a library of solid models of standard parts with optional sizes is not available (0).

### 3.1.6. Projection Theory/Projection of Multi-views from 3-D Models

R6.1. Create multi-views from 3-D model and keep them associative

A6.1. Creating multi-views from 3-D model is possible. Changes in the model are reflected to layout page automatically, but reverse is not possible, i.e. model-multiview data associativity is unidirectional as shown in FIGURE 25.

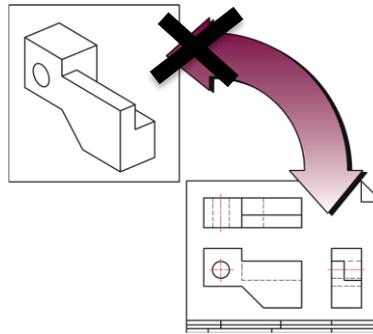


FIGURE 25. Model-data associativity [19].

R6.1.1. Create multi-views from 3-D model and keep solid model-drawing data associative

A6.1.1. Creating multi-views from 3-D model is possible. Any change in solid model is automatically reflected to multi-views created in layout page (1).

R6.1.2. Keep drawing data - model associative

A6.1.2. After creating multi-views from 3-D model, changing the solid model through modifying multi-views in layout page is not possible with AutoCAD Mechanical (0).

R6.2. Create perspective and isometric projections

A6.2. One-point perspective, and isometric projection can be obtained from 3-D model automatically. Principal views can be obtained for both third-angle projection and first-angle projection as shown in FIGURE 26 (1).

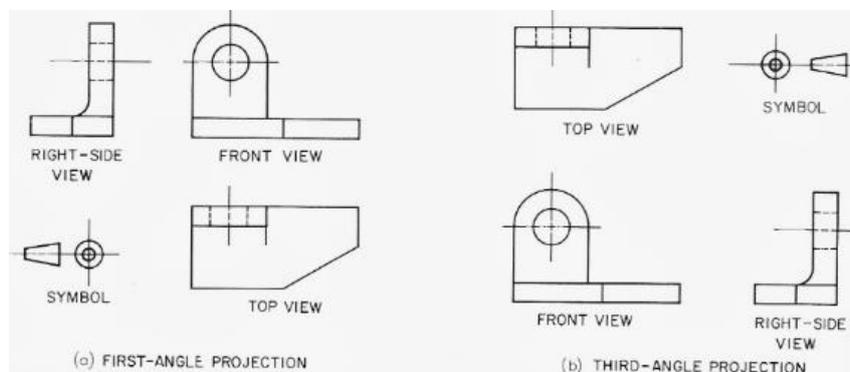


FIGURE 26. Third-angle projection and first-angle projection [21].

### 3.1.7. Projection of Third Principal View from Other Views/Auxiliary View

R7.1. Align the principal views

A7.1. Three principal views can be aligned by using object snapping, miter line and construction lines as shown in FIGURE 27 (1).

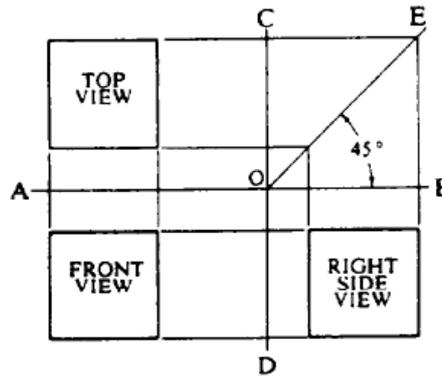


FIGURE 27. Alignment of principal views.

R7.2. Complete the missing lines

A7.2. Students should do exercises to improve their visualization capabilities. Some incomplete drawings with missing lines are given and the students are expected to complete the missing lines. AutoCAD is a flexible tool in such exercises like the one given in FIGURE 28 (1).

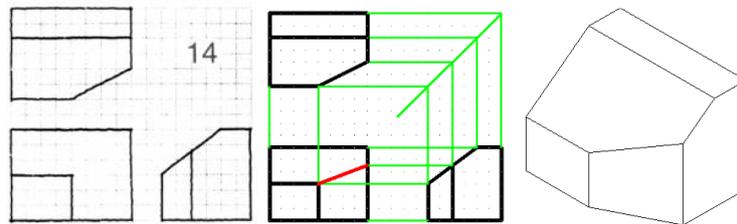


FIGURE 28. Missing line exercise.

R7.3. Draw the missing view

A7.3. Multi-view drawings with a missing view are given to the students and they are expected to draw the missing view like the one shown in FIGURE 29 (1).

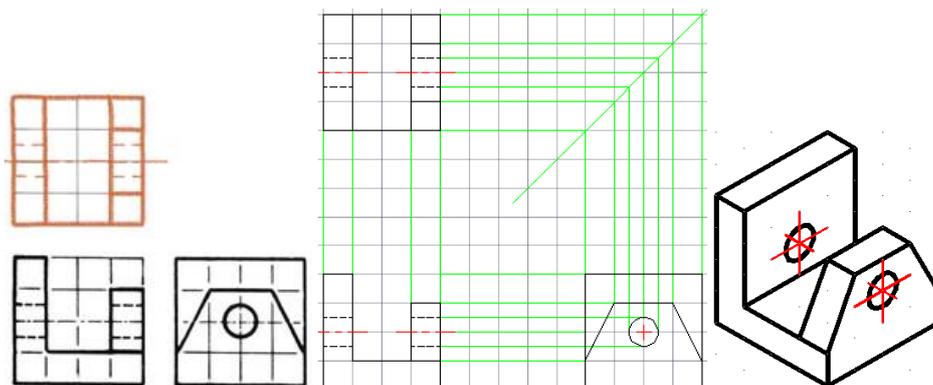


FIGURE 29. Missing view exercise.

#### R7.4. Create auxiliary views

A7.4. An auxiliary view from a solid model can be easily created by looking at the inclined or oblique face of model at right angle. When the multi-views are used; primary, secondary, tertiary auxiliary view can be drawn by aligning coordinate axes parallel and perpendicular to the related edge view, using object snapping, and measuring and locating the distances of the related features from the fold lines. An example is given in FIGURE 30 (1).

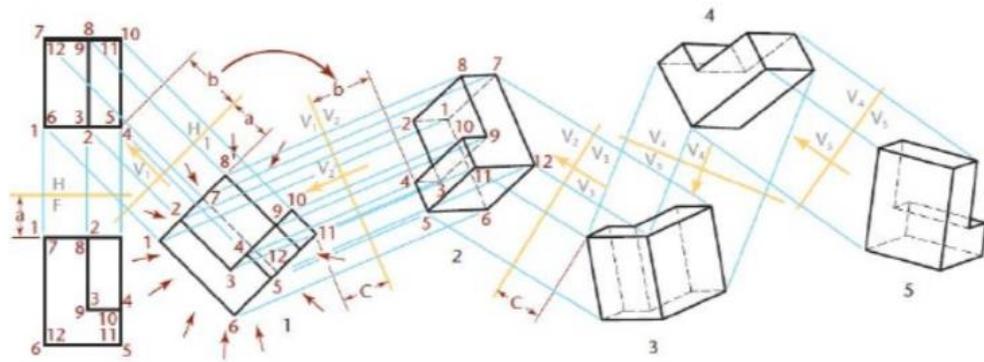


FIGURE 30. Auxiliary views [22].

#### R7.5. Draw a point view of a line

A7.5. The point view of a line can easily be drawn with solid models by looking at the model in the direction of the line. In the case of multi-views, the view can be obtained in two steps: first drawing primary auxiliary view by looking at the line at right angle, then drawing secondary auxiliary view through looking at the line of primary auxiliary view in the direction of the line (1).

#### R7.6. Measure a dihedral angle

A7.6. Keeping the line of sight parallel to the edges of the angular surfaces, and then drawing the primary auxiliary view gives the true angle auxiliary view. In solid model, it can be measured directly by drawing two intersecting lines that are perpendicular to the edges of the inclined faces and measuring the angle between these two intersecting lines (1).

#### R7.7. Draw true size of an oblique surface

A7.7. It can be drawn through constructing successive auxiliary views in the way shown in FIGURE 31 (2).

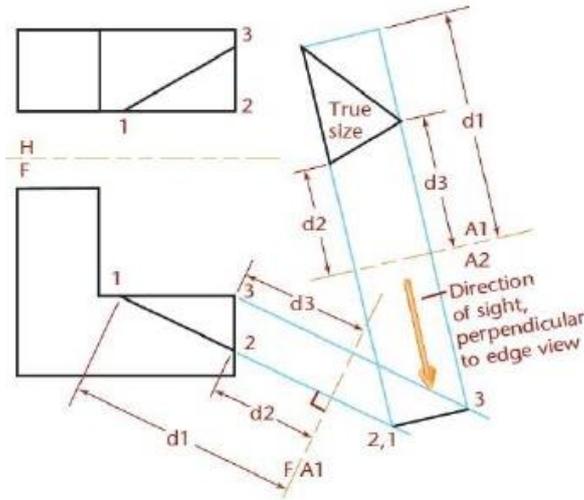


FIGURE 31. True size of an oblique surface.

### 3.1.8. Pictorial Projection/Isometric Drawing

R8.1. Create isometric drawing of an object

A8.1. Isometric drawing can be created in two ways: automatic creation from a solid model or drawing it in 2-D using isometric axes and grid system. Regular and reversed axis isometric are easy to draw, however long axis isometric requires rotation of isometric axis. Using the proper isometric plane and ‘ortho’ mode, isometric drawing is easy to create. Steps of drawing are shown in FIGURE 32 (1).

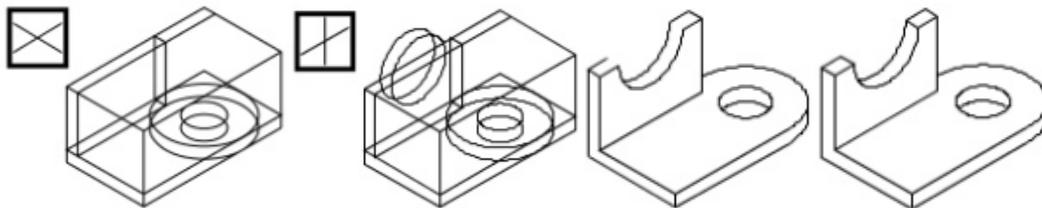


FIGURE 32. Isometric drawing in 2-D [23].

R8.2. Draw an isocircle

A8.2. It is easy to draw an isocircle after shifting to the proper isometric plane, and using isocircle option of ‘ellipse’ command. However, rounding the corners is not possible with ‘fillet’ command, and it requires first drawing an isocircle on the right isoplane and then trimming unnecessary portions (1).





#### R9.4. Draw an offset section view

A9.4. An offset section view can be created automatically from solid model or be drawn in 2-D as given in FIGURE 37 (1).

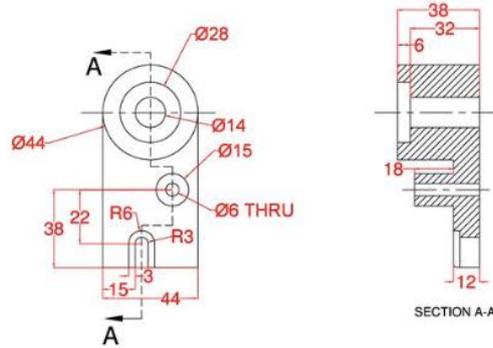


FIGURE 37. Offset section view.

#### R9.5. Create a rotated (revolved) section view

A9.5. There is not any command for a rotated section view, but it can be obtained as full section view and then move it to the required position. It is possible in 2-D as shown in FIGURE 38.

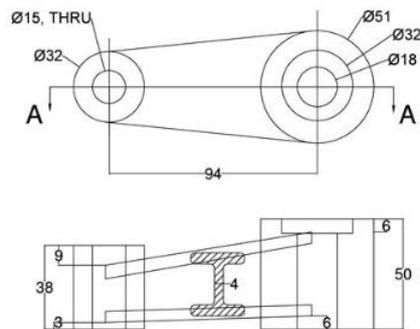


FIGURE 38. Revolved section view.

##### R9.5.1. Create a rotated (revolved) section view from solid

A9.5.1. There is no special command. However, it can be obtained as full section view be moved to the required location (1).

##### R9.5.2. Create a rotated (revolved) section view in 2-D

A9.5.2. Drawing a revolved section view in 2-D is possible (1).

#### R9.6. Section ribs, webs, lugs and other thin features

A9.6. Automatic section lining (cross hatching) from a solid model is not possible. However, they can be arranged in 2-D section views. An example is given in FIGURE 39.

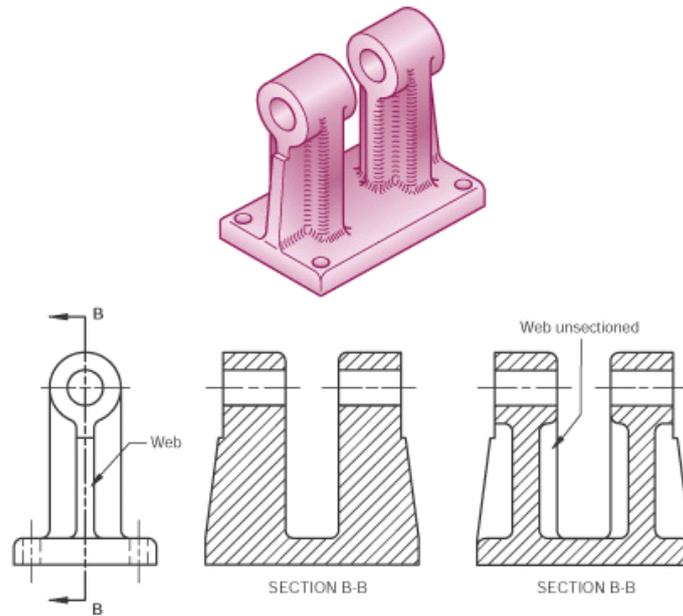


FIGURE 39. Section view of a solid model [25].

R9.6.1. Section ribs, webs, lugs and other thin features from a solid model

A9.6.1. Automatic section lining from a solid model is not possible since rib, web or lug is not defined as solid. Spoke and web discrimination is not made (0).

R9.6.2. Section ribs, webs, lugs and other thin features in 2-D

A9.6.2. Drawing section view and omitting hatching in 2-D is possible (1).

R9.7. Create an aligned section view

A9.7. Aligned section view is possible both in 3-D and 2-D as shown in FIGURE 40.

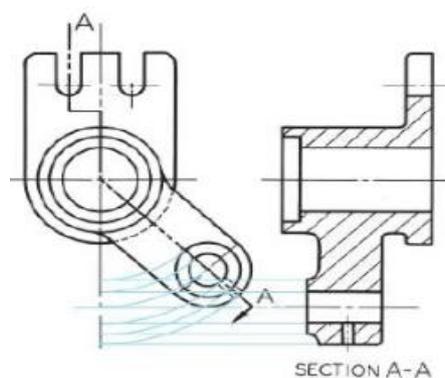


FIGURE 40. Aligned section view.

R9.7.1. Create an aligned section view from a solid model

A9.7.1. AutoCAD provides foreshortened true projection in full section view. However, rotated conventional aligned section view can be obtained from the solid model by creating aligned cutting plane lines (1).

R9.7.2. Create an aligned section view in 2-D

A9.7.2. Rotated conventional aligned section view is possible in 2-D (1).

R9.8. Draw a broken-out section view

A9.8. It is possible to draw a broken-out section view in both 2-D and 3-D. An example is given in FIGURE 41 (1).

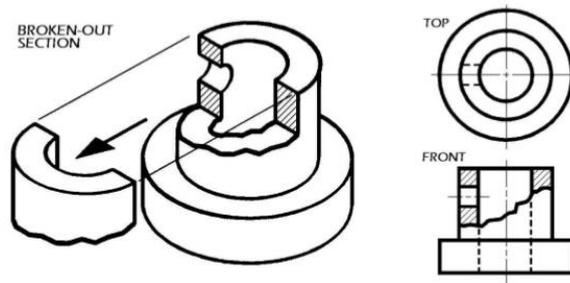


FIGURE 41. Broken-out section view [26].

### 3.1.10. Dimensioning and Tolerancing

R10.1. Dimension and assign tolerances in accordance with the international standards

A10.1. AutoCAD provides the dimensioning and tolerancing tools that are in line with international standards. Basic standards are given on tolerancing and dimensioning in [27] and [28], on limits and fits in [29] and [30], on geometrical tolerances in [31], [32] and [33] (1).

R10.2. Use unidirectional or aligned system of placing dimensions

A10.2. Unidirectional is standard for mechanical engineering applications. Both systems of placing can be used as shown in FIGURE 42 (1).



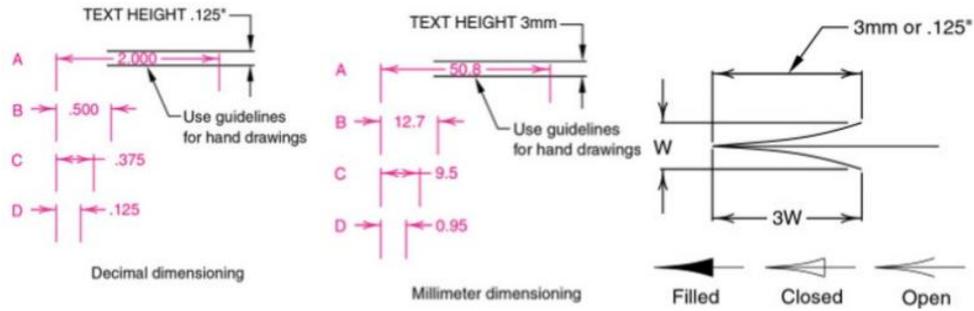


FIGURE 44. Standard sizes for dimensional text and lines [34].

R10.5. Dimension the drawing using datum (baseline), ordinate or linear continued dimensioning

A10.5. AutoCAD provides alternative dimensioning methods. An example of baseline dimensioning is given in FIGURE 45 (1).

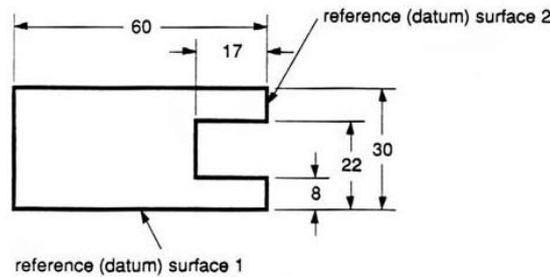


FIGURE 45. Baseline dimensioning example.

R10.6. Create basic and reference dimensions

A10.6. Basic or reference dimensioning is possible. An example is shown in FIGURE 46 (1).

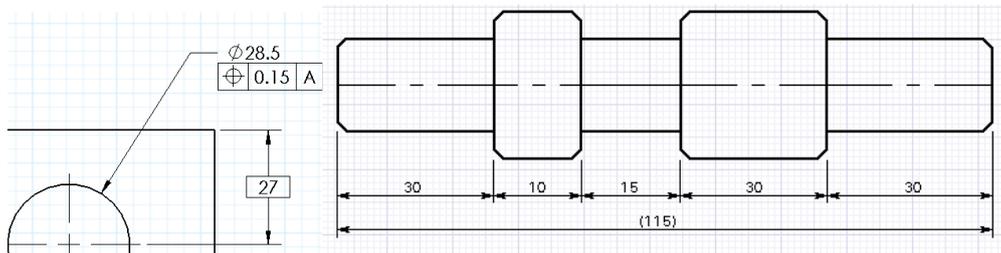


FIGURE 46. Basic and reference dimensions.

R10.7. Define dimension units

A10.7. SI or US customary units can be used. Angular dimensions can be defined as decimal degree or degrees, minutes, seconds (1).

R10.8. Dimension linear and aligned features

A10.8. It is easy with object snapping feature at the desired location (1).

R10.9. Dimension arcs

A10.9. Arcs are dimensioned easily by using object snapping feature. An example is given in FIGURE 47 (1).

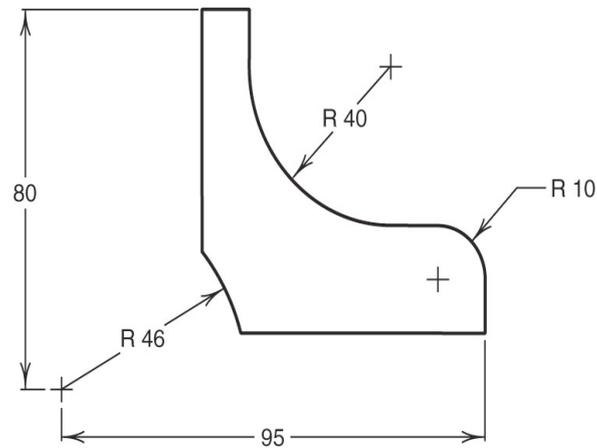


FIGURE 47. Arc dimensioning example.

R10.10. Dimension angular features

A10.10. It requires selecting lines of the angle. It can be placed at the proper location (1).

R10.11. Dimension a circle and repetitive features

A10.11. A circle can easily be dimensioned. Repetitive features can be dimensioned using X as shown in FIGURE 48 (1).

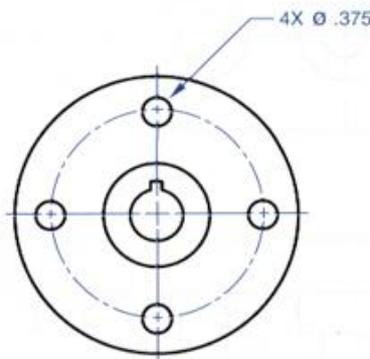


FIGURE 48. Dimensioning a circle and repetitive features.

R10.12. Show drilling operations on the drawing

A10.12. Counter bore, countersink and spot face features can be defined easily as shown in FIGURE 49 (1).

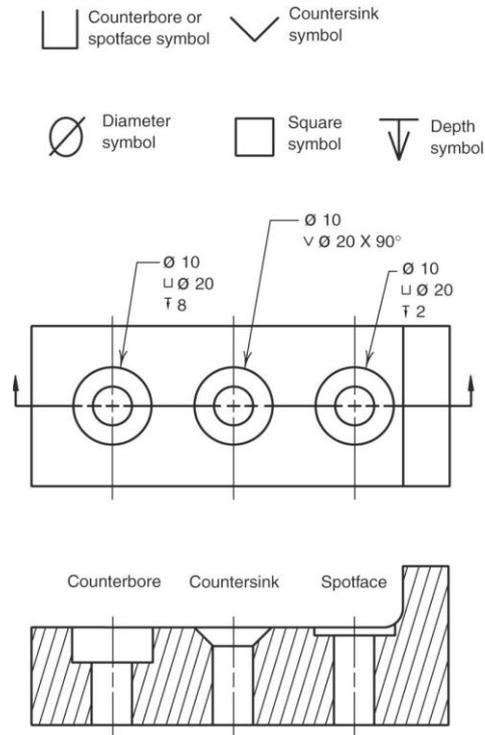


FIGURE 49. Counter bore, countersink and spot face dimensioning.

R10.13. Draw center marks and center lines

A10.13. It is practical to draw center mark or center lines through selecting circular feature (1).

R10.14. Assign tolerances to a drawing

A10.14. Tolerances can be expressed in several ways such as direct limits or as tolerance values, geometric tolerances, notes referring to specific conditions, a general tolerance note in the title block (1).

R10.15. Show surface texture

A10.15. Surface texture information such as average surface roughness height, roughness width, roughness width cutoff, lay direction, waviness height, waviness width, information on material removal can be shown in ISO or ANSI format. An example is given in FIGURE 50 (1).

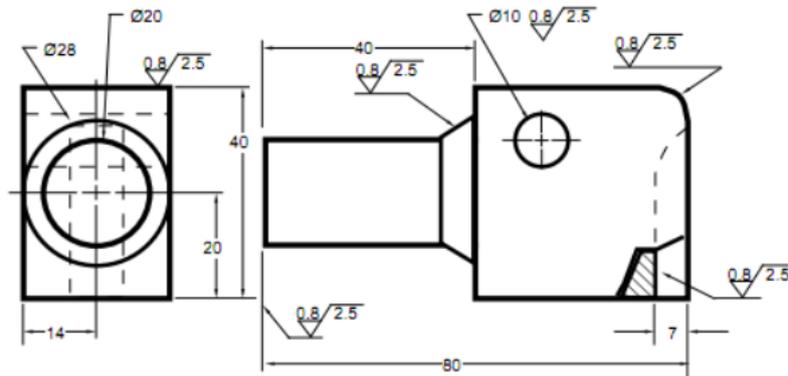


FIGURE 50. Surface texture

**3.1.11. Basics of Assembly Drawing**

R11.1. Create an assembly drawing in 2-D

A11.1. Assembled assembly drawing is easy to create using the general drawing tools. AutoCAD has a 2-D library of standard parts, and they can be used in assembly drawing. Exploded assembly drawing, especially isometric drawing of standard parts takes much time than the assembled. Examples of the assembly drawing are given in FIGURE 51 (1).

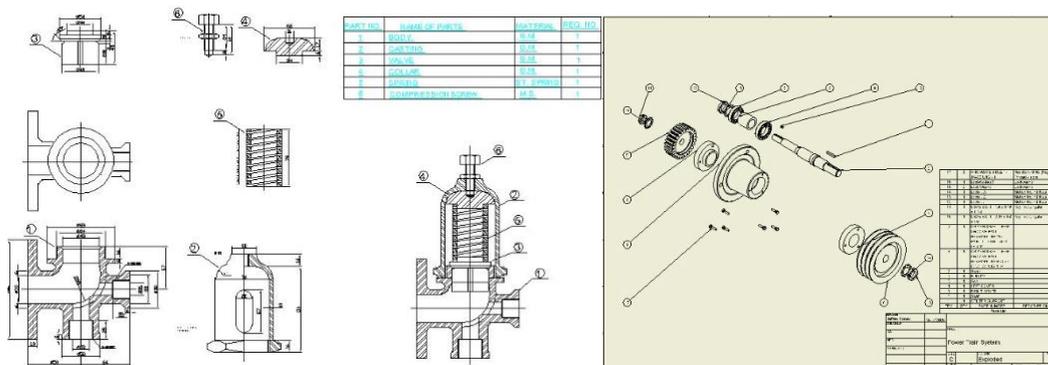


FIGURE 51. Assembly drawing: assembled and exploded.

R11.2. Create an assembly drawing in 3-D

A11.2. Although AutoCAD is a software created mainly for 2-D drawings, its capabilities has been extending into 3-D drawings. Creating assembly drawings in 3-D is possible, but takes much time in comparison to solid modelling software (1).

R11.3. Construct an assembly by using the parts' available solid models

A11.3. Starting with a base component and adding other components using geometric relations is the way used. Mating and aligning are the most common methods for relating parts to each other in the assembly, however mating is handled indirectly (1).

R11.4. Create a parts list in assembled and exploded assembly drawings

A11.4. Parts list or bill of materials can be created. However, different numbers are assigned to the same part in assembled and exploded assembly views.

R11.4.1. Create a parts list

A11.4.1. Parts list or bill of materials is easy to create (1).

R11.4.2. Assign the same number for the same part in the parts list

A11.4.2. AutoCAD does not differentiate multiple instances of the same part in creating assembled and exploded views of the assembly in the same drawing file, and therefore different numbers are assigned for the same part in assembled and exploded assembly drawings (0).

R11.5. Create a pictorial assembly drawing

A11.5. It can be created both in 2-D and 3-D (1).

R11.6. Create an outline assembly drawing

A11.6. Outline assembly can be drawn in 2-D and 3-D (1).

R11.7. Create a sectioned assembly drawing

A11.7. It can be created as pictorial or multi-view form. Pictorial form can be achieved easily through solid modelling, however it takes much time to draw it in 2-D. Sectioning in multi-view can be done in 2-D and 3-D. Interior features of an assembly can be shown by using also a different technique such as transparency (1).

### **3.1.12. Presentation Techniques of Engineering Drawings/Printing**

R12.1. Create a printout of a drawing

A12.1. Printouts can be taken in general from the layout page, but in exceptional cases from the model space. The layout page can be prepared in different page sizes using the templates which are in line with international and company standards. Many layout pages can be prepared from a single model (1).

R12.2. Use rendering techniques to create artistic drawings

A12.2. An introductory information on rendering techniques is given to the students, although creating artistic images is out of scope of mechanical engineering education. AutoCAD

provides some tools in rendering, such as light source, shadow, sun status and location, material, texture and mapping (1).

### 3.1.13. Results of the Assessment

The overall results of the assessment of AutoCAD Mechanical in meeting the requirements of CAD I course are summarized in TABLE 6. 96 of 104 educational requirements are fulfilled by AutoCAD Mechanical. The overall rating of AutoCAD in terms of fulfilment of educational requirements is 92.3%.

Relatively low rate of meeting the requirements are given in solid modelling fundamentals (13 of 17), obtaining multi-views from 3-D model (2 of 3), pictorial projection (5 of 6) and basics of assembly drawing (7 of 8).

TABLE 6. Assessment of AutoCAD in terms of meeting CAD I requirements.

Number	Content	No of requirements	No of fulfilled requirements	Percent of fulfillments
1	Basics of engineering drawing	3	3	100
2	Coordinate systems	4	4	100
3	Basic geometric elements	7	7	100
4	Drawing simple geometric objects and editing	21	21	100
5	Solid modelling fundamentals	17	13	76.5
6	Projection theory/multi-views from 3-D	3	2	66.7
7	Projection of 3 <sup>rd</sup> view from others	7	7	100
8	Pictorial projection	6	5	83.3
9	Section view	11	10	90.9
10	Dimensioning and tolerancing	15	15	100
11	Basics of assembly drawing	8	7	87.5
12	Presentation of engineering drawings/printing	2	2	100
<b>Overall</b>		<b>104</b>	<b>96</b>	<b>92.3</b>

### 3.2. Assessment of AutoCAD in Terms of Fulfillment of the Requirements of a CAD II Course

The second Computer Aided Drawing/Design (CAD) course (CAD II) deals mainly with assembly drawing and descriptive geometry.

Similar to the CAD I assessment, the main subjects of the course CAD II, the requirements, and assessment of AutoCAD in meeting the requirements are evaluated. Each numbered item defines a main subject of the course, while R and A stand for the requirements in that area and assessment of AutoCAD in fulfilling the requirement, respectively.

### 3.2.1. Introduction to Solid Modelling

A general assessment in part drawing is made in the CAD I assessment. Additional points are given below.

#### R1.1. Create basic surfaces

A1.1. The '3dface' command provides triangular or quadrilateral flat (planar) surfaces from the points specified. The points can be selected on the display by using the object snap facility or entering their absolute or relative coordinates (1).

#### R1.2. Create complex surfaces by using the sweeping tool

A1.2. Swept surfaces can be generated by sweeping generator entities along director entities. An example application is given in FIGURE 52.

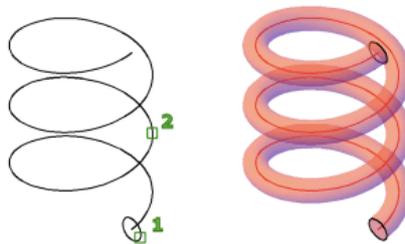


FIGURE 52. Sweep command application.

A more complex surface can be created by sweeping directrix along a curved generatrix, as shown in FIGURE 53 (1).

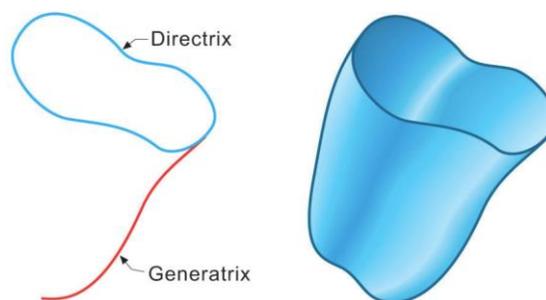


FIGURE 53. Sweeping directrix along generatrix [18].

#### R1.3. Create complex surfaces by using revolve tool

A1.3. Select a line, arc, circle, or 2D or 3D polyline to sweep in a circular path around a selected axis. 'revsurf' command is used. An example is given in FIGURE 54 (1).

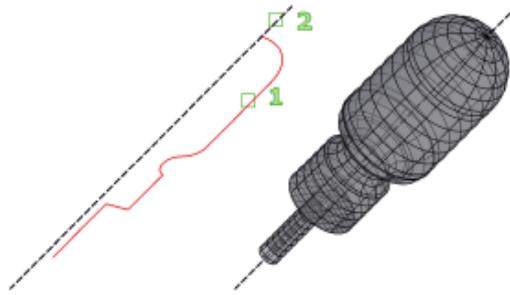


FIGURE 54. Revsurf command application.

#### R1.4. Create complex surfaces or solids by using loft tool

A1.4. With the 'loft' command, a new solid or surface can be created by specifying a series of cross sections. The cross sections define the profile (shape) of the resulting solid or surface. Cross sections (generally, curves or lines) can be open (for example, an arc) or closed (for example, a circle). An example application is shown in Figure FIGURE 55 (1).

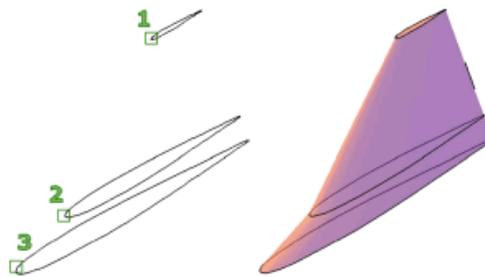


FIGURE 55. Loft command application.

#### R1.5. Constrain profile geometry for 3-D modelling

A1.5. Constraints can be created in 2-D by using geometric and dimensional constraints. The drawing shown in FIGURE 56 can be drawn in 2-D, but it loses all the constraints after creating the third dimension. It is not possible with AutoCAD to define constrains in 3-D. This is a major shortcoming of AutoCAD. Third dimension can be obtained after changing the parameters in 2-D. Another deficiency is the unavailability of Offset constraint (0).

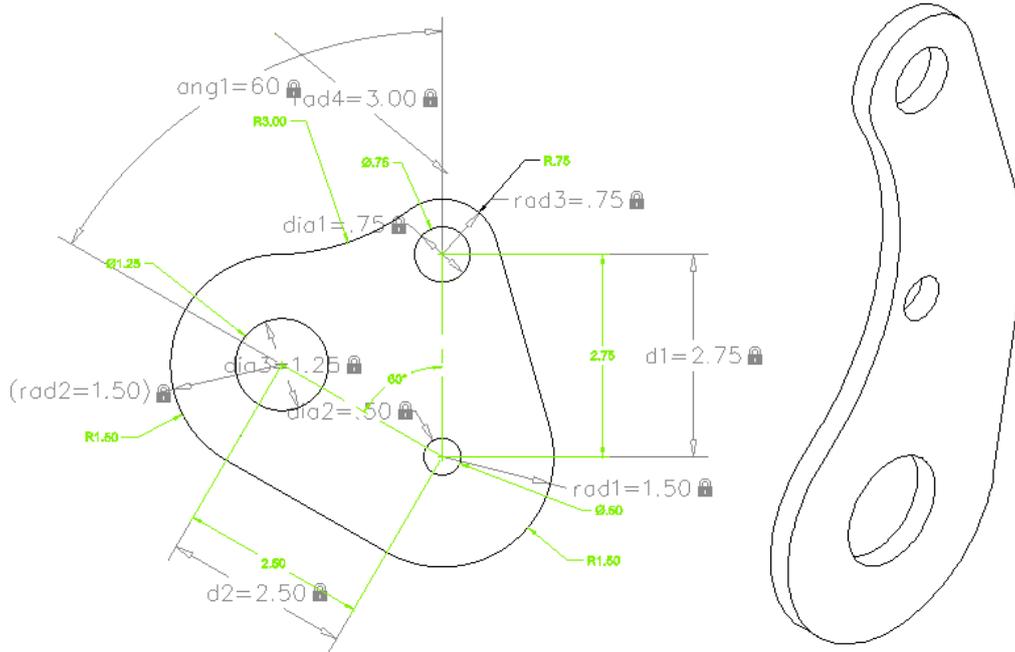


FIGURE 56. Geometric and dimensional constraining example.

**3.2.2. Construction of Solid Features**

R2.1. Create standard solid features

A2.1 The solid feature dialogue box like the one given in FIGURE 57 is not available with AutoCAD Mechanical (0).

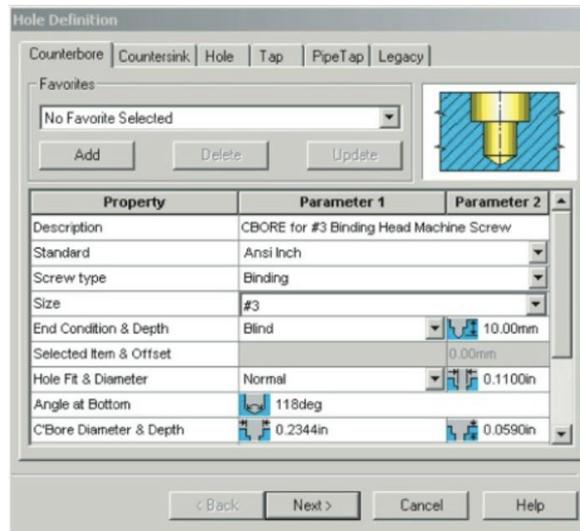


FIGURE 57. Solid features dialogue box example.

### R2.2. Edit part feature tree

A2.2. AutoCAD does not recognize a solid as a part, therefore part feature cannot be changed using the feature tree, which is similar to the one shown in FIGURE 58. However, a properties table is available with some changeable features (0).

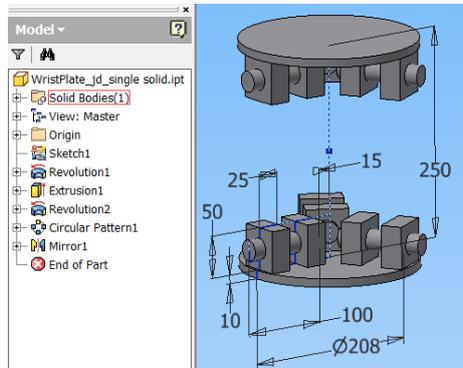


FIGURE 58. Part feature tree example.

### R2.3. Create feature-based solid models

A2.3. Feature-based modeling refers to the construction of geometries as a combination of form features. Solid model of a part can be obtained from the raw material (workpiece) through form features as shown in the example application given in FIGURE 59 (1).

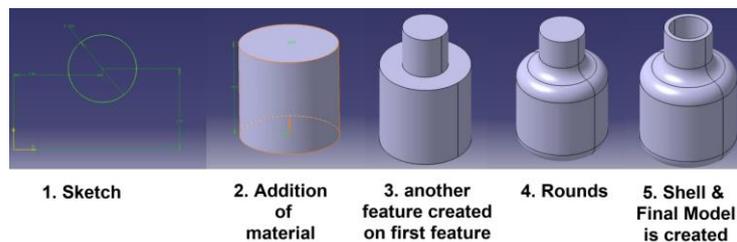


FIGURE 59. Feature-based modeling example [35].

### R2.4. Create a rationale modelling with algebraic equations

A2.4. An equation driven solid model can be obtained from the multi-views drawn using parametric algebraic equations as shown in FIGURE 60. Since constraints are deleted after solid modelling, parameters are changed before creating solid model from the multi-views.

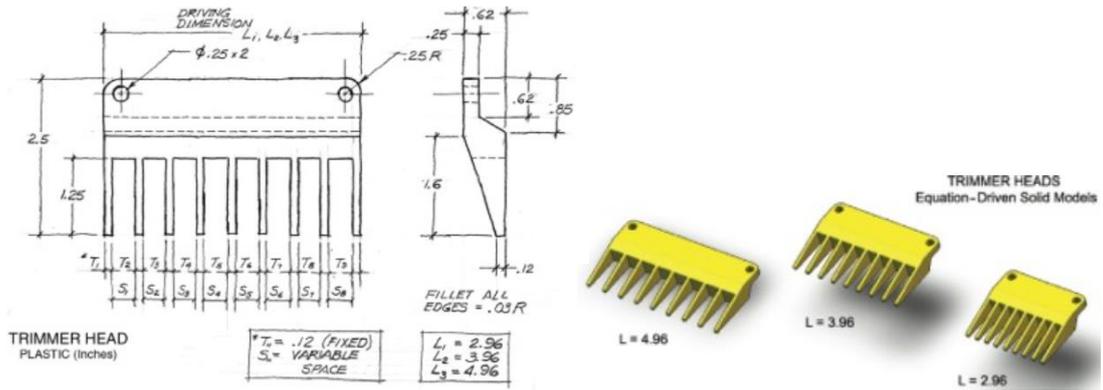


FIGURE 60. Equation driven solid model example [18].

R2.4.1. Create a rationale modelling with algebraic equations in 2-D

A2.4.1. Equation driven solid model can be obtained in 2-D (1).

R2.4.2. Create a rationale modelling with algebraic equations in 3-D

A2.4.2. Equation driven solid model in 2-D is not possible with AutoCAD Mechanical (0).

**3.2.3. Tolerances, Surface Quality Marks**

The general assessment on tolerances is made in the CAD I context. The assessment for assembly drawings is given below.

R3.1. Describe the nominal size, tolerance, limits, and allowance of two mating parts.

A3.1 Values for maximum and minimum allowable sizes expressed as maximum material condition (MMC) and least material condition (LMC) can easily be assigned. Nominal size, tolerance, limits, allowance of mating parts can be shown. An example of limit values is shown in FIGURE 61 (1).

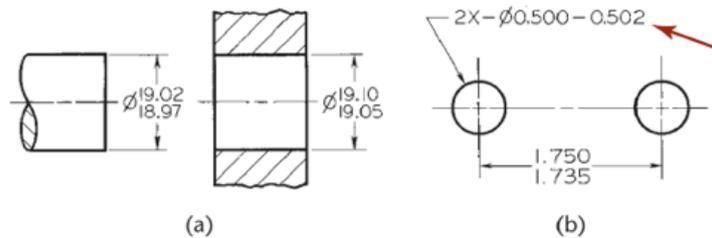


FIGURE 61. Limit tolerancing example.

R3.2. Identify a clearance fit, interference fit, and transition fit

A3.2. Fitting data of mating parts can be assigned, and be shown in the desired format as shown in FIGURE 62. ISO or ANSI standards can be used in designation. In ISO; clearance,

transition and interference fits, in ANSI; running and sliding, clearance locational, transition locational, interference locational, and force and shrink fits can be defined.

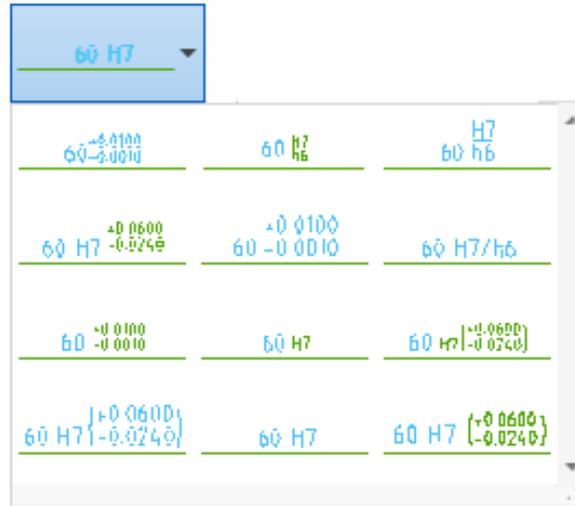


FIGURE 62. Options for fit format.

R3.2.1. Show clearance fit on drawing

A3.2.1. Clearance fit in ISO or ANSI format can be identified easily (1).

R3.2.2. Identify transition fit on drawing

A3.2.2. Transition fit can be defined and shown on engineering drawing in ISO or ANSI format (1).

R3.2.3. Show interference fit on drawing

A3.2.3. Interference fit in ISO or ANSI format can be identified easily (1).

R3.3. Describe the basic hole and basic shaft systems

A3.3. Hole or shaft basis fit, and assigned tolerances with the basic size of minimum hole or maximum shaft size can be defined using the ISO and ANSI standard values. An example is given in FIGURE 63.

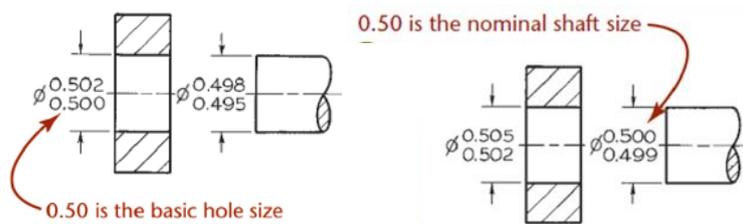


FIGURE 63. Hole and shaft basis fit examples.

R3.3.1. Describe the mating parts by using the hole system

A3.3.1. Hole basis fitting can easily be defined (1).

R3.3.2. Describe shaft basis fitting

A3.3.2. AutoCAD has a data base to define ISO or ANSI standard, shaft or hole basis fitting (1).

R3.4. Dimension mating parts by using limit dimensions, unilateral tolerances, and bilateral tolerances.

A3.4. There are many options in dimensioning and tolerancing of mating parts. Basic size, fit type with fundamental deviation and international grade data, limit values and tolerances can be shown. Limit form or note form can be used as explained in FIGURE 64 (1).

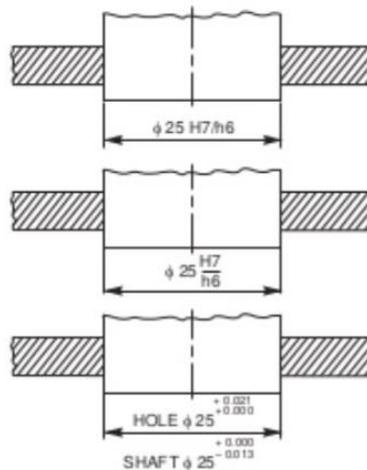


FIGURE 64. Example for limit form or note form of metric tolerancing.

### 3.2.4. Geometric Dimensioning and Tolerancing (GDT)

R4.1. Use feature control frame to assign GDT symbols

A4.1. Feature control frames with the required GDT symbols can be added as shown in the example of FIGURE 65 (1).

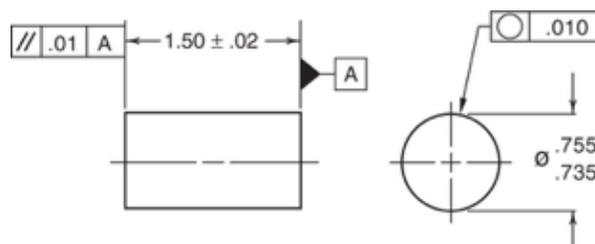


FIGURE 65. Feature control frame example.

R4.2. Create datum and datum features

A4.2. Datum and datum features can be created. An example of datum and feature control frame application is given in FIGURE 66 (1).

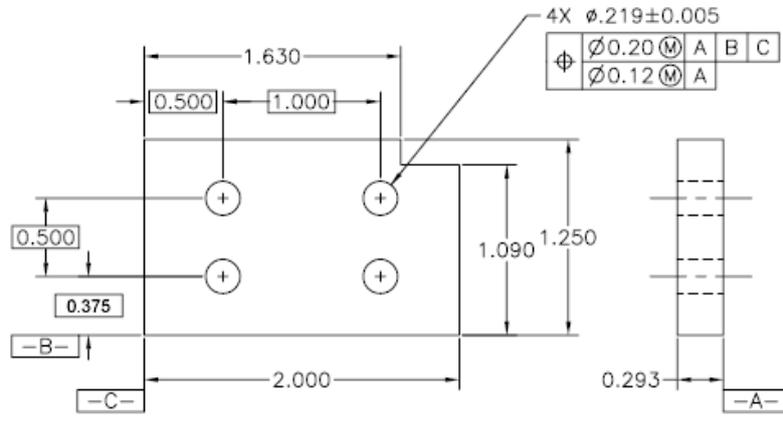


FIGURE 66. Datum feature and feature control frame example.

R4.3. Define material quality conditions

A4.3. Material quality values of maximum material condition (MMC, M), least material condition (LMC, L), and regardless of feature size (RFS, S) can be assigned by using feature control frame as shown in FIGURE 67 (1).

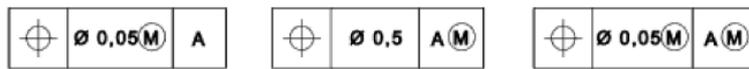


FIGURE 67. Material quality condition examples.

R4.4. Define complex tolerances

A4.4. Complex tolerancing data can assigned by using feature control frames like the one given in FIGURE 68 (1).

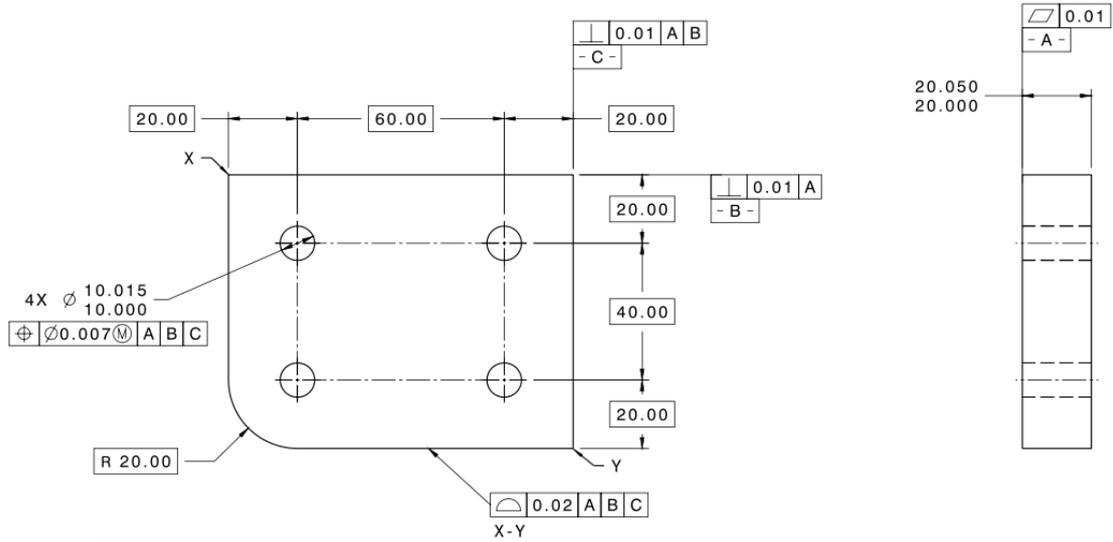


FIGURE 68. Complex tolerancing example.

R4.5. Define form controls such as straightness, flatness, circularity, cylindricity

A4.5. Form controls can be defined. An example is given in FIGURE 69 (1).

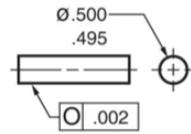


FIGURE 69. An example of circularity control.

R4.6. Define orientation controls such as parallelism, perpendicularity, and angularity.

A4.6. Orientation controls can be defined. An example is given in FIGURE 70 for perpendicularity control (1).

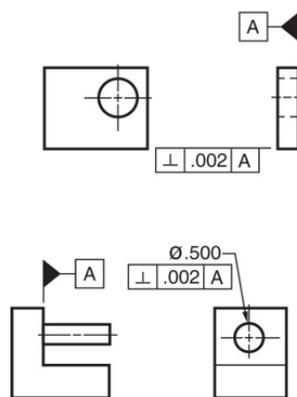


FIGURE 70. An example of perpendicularity control.

R4.7. Define profile controls such as line profile and surface profile.

A4.7. Profile controls can be defined. An example is given in FIGURE 71 for surface profile control (1).

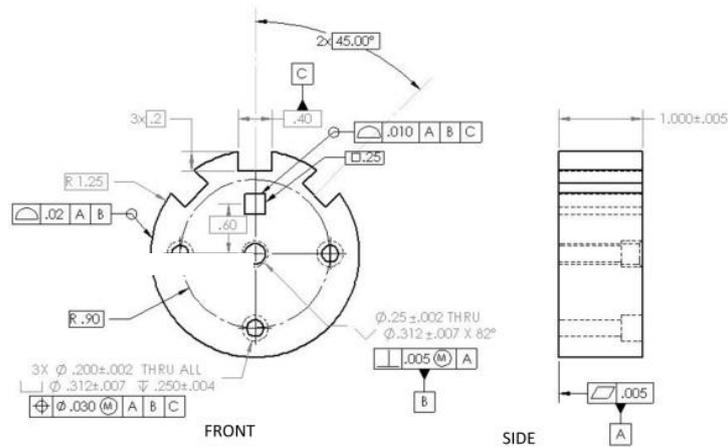


FIGURE 71. An example of surface profile control.

R4.8. Define location controls such as concentricity, position, and runout.

A4.8. Location controls can be defined. An example is given in FIGURE 72 for circular runout control (1).

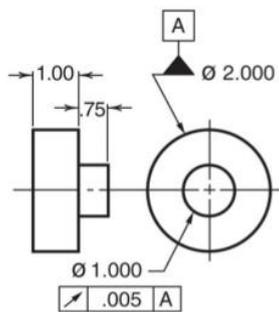


FIGURE 72. An example of circular runout control.

### 3.2.5. Working Drawing and Assemblies

The general assessment on assembly drawing is made in the CAD I context. An additional assessment is given below.

R5.1. Create revision registers

A5.1. Revision blocks can be created and changes in assembly drawings and part drawings can be registered (1).

### 3.2.6. Threaded Fasteners

#### R6.1. Draw threads in 2-D

A6.1. Bolt, stud, and screw drawings can be created in detailed, schematic and simplified forms in accordance with international standards. They can be drawn using AutoCAD content library or available drawing tools as shown in FIGURE 73. Thread notes should be added by the user (1).

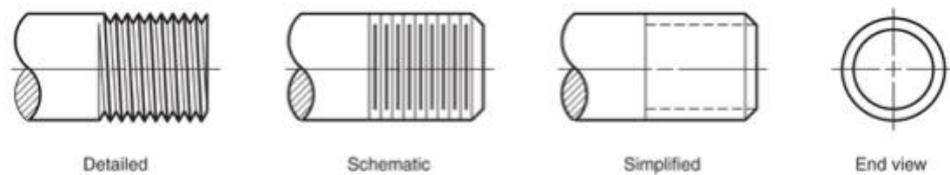


FIGURE 73. Thread drawing example with different options.

#### R6.2. Draw threads in 3-D

A6.2. It is possible but takes much time to draw solid model of standard threads. However, AutoCAD does not recognize the model as a standard threaded part. Therefore, automatic 2-D projection from the solid model is not in accordance with international standards (0).

### 3.2.7. Non-Threaded Fasteners, Springs

#### R7.1. Draw non-threaded fasteners in 2-D

A7.1. Pin, key, rivet, retaining ring, and washer drawings can be created in accordance with international standards. They can be drawn using AutoCAD library or available drawing tools as shown in FIGURE 74. AutoCAD library does not include all types. Designation notes should be added by the user (1).

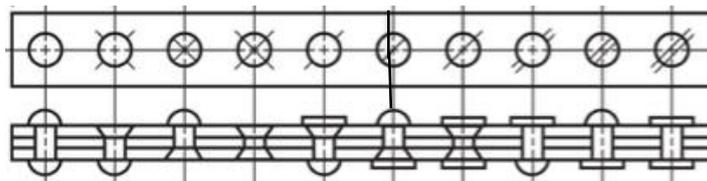


FIGURE 74. Rivet examples.

#### R7.2. Draw non-threaded fasteners in 3-D

A7.2. It is possible to create solid model by using either content library 2-D sketch or solid modelling techniques (1).

### R7.3. Draw spring

A7.3. Spring drawings can be created in 2-D using library items or directly drawing using the available 2-D tools. 3-D drawing can be obtained from the manufacturer or be drawn by using 3-D drawing tools. Creating 2-D projection from the 3-D drawing is possible for detail drawing (1).

## **3.2.8. Mechanisms**

### R8.1. Draw a spur, worm, and bevel gear in 2-D

A8.1. Gear drawings can be created using library items or by using the drawing tools. Labeling in accordance with ANSI or ISO standards is possible. Shafts, keyways, and key seats can be drawn. While using library item, automatic creating of other views from the available view is possible. Gear cutting data can be shown.

#### R8.1.1. Draw a spur gear in 2-D

A8.1.1. Spur gear is easy to draw by using the 2-D content library. Non-standard spur gear can be drawn using 2-D drawing tools (1).

#### R8.1.2. Draw a worm gear in 2-D

A8.1.2. It is easy to draw a worm gear by using the 2-D content library. It is also possible to draw by using 2-D drawing tools (1).

#### R8.1.3. Draw a bevel gear in 2-D

A8.1.3. Bevel gear can be drawn by using the 2-D content library. An alternative to the content library is to draw by using 2-D drawing tools (1).

### R8.2. Draw a spur, worm, and bevel gear in 3-D

A8.2. Creating solid model from an available content library of 2-D gear drawings is easy. It is also possible to draw using the 2-D drawing tools, but it takes much time in comparison to drawing by using 2-D content library (1).

### R8.3. Draw two gears in mesh in 2-D

A8.3. It is possible to create two spur or bevel gears in mesh. An example is shown in FIGURE 75 (1).

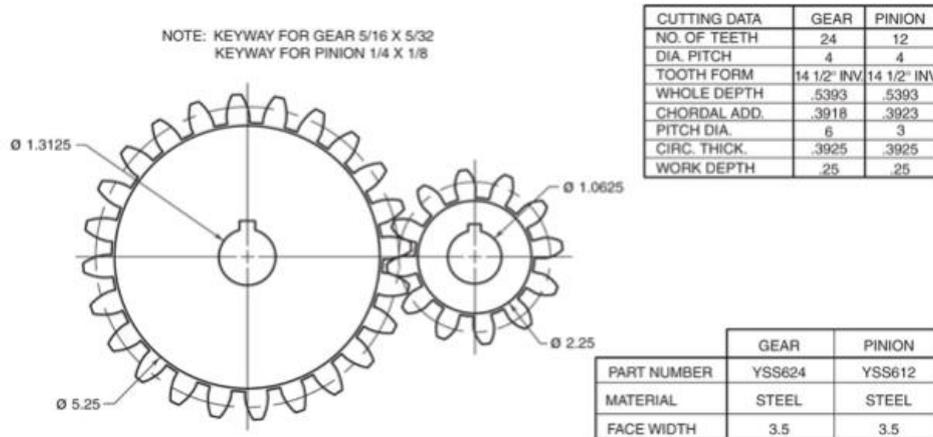


FIGURE 75. Example of two spur gears in mesh.

R8.4. Draw a cam profile, follower, and displacement diagram in 2-D.

A8.4. AutoCAD has a wizard to create a cam profile, follower and displacement diagram. However, the number of options for displacement is limited. An example drawing of cam and follower is given in FIGURE 76. Creating the drawing using 2-D drawing tools is also possible (1).

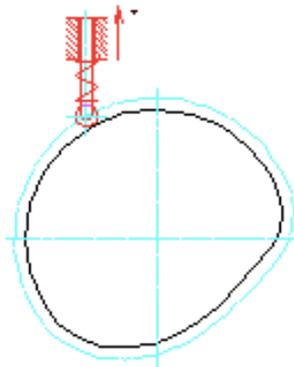


FIGURE 76. Cam profile example

R8.5. Create a standard bearing drawing in 2-D.

A8.5. Standard bearing drawings can be created using AutoCAD content library (1).

### 3.2.9. Welding

R9.1. Designate welding using standard weld symbols

A9.1. AutoCAD provides ISO and ANSI standard symbols to designate all the data related to welding such as reference line, leader line and arrow, basic weld symbol (location, depth of

weld), finish symbol, weld symbol (type of weld), dimensions, supplementary symbols, tail and specifications (1).

### **3.2.10. Descriptive Geometry**

R10.1. Create an auxiliary view using reverse construction technique

A10.1. It can be created using the geometric data of the true geometry in 2-D or 3-D (1).

R10.2. Find true length of a line

A10.2. It can be obtained by creating required auxiliary views, using snapping and aligning coordinate axes (1).

R10.3. Create point view of a line

A10.3. It is obtained when it is looked at the object in the line direction. This view is created by drawing successive auxiliary views (1).

R10.4. Create true shape and size of a plane

A10.4. True shape of a plane can be obtained by creating an auxiliary orthographic view parallel to the plane. This is obtained with the second auxiliary view (1).

R10.5. Create the geometry to be obtained at the intersection of two bodies

A10.5. It is very easy with 3-D modelling. Boolean operator 'intersection' creates the intersection of two bodies (1).

R10.6. Locate a line or a point on a plane

A10.6. 2D intersection points of the line with edges are defined in one of the principal views. The intersection points are projected on the other view, and the points are joined with a line. The intersection points are extended to end points. The points of intersection with vertical lines are found. To locate a point on a plane, a line is drawn from one of the plane's corner points through the point. The line on the other view is located (1).

R10.7. Determine piercing points

A10.7. Piercing points can be determined by locating intersecting points and projecting intersections, after drawing two principal views of the planes and the lines (1).

R10.8. Determine shortest distance between a line and point

A10.8. Using two principal views of the line and point, the distance between the two points that are obtained after getting secondary auxiliary view is measured (1).

R10.9. Determine distance between two lines

A10.9. Using two principal views of the lines, the distance between the point and the line that are obtained after the secondary auxiliary view is measured. It is very easy in 3-D (1).

### 3.2.11. Results of the Assessment

The results of the assessment of AutoCAD Mechanical in meeting the requirements of CAD II course are summarized in TABLE 7. 63 of 73 educational requirements are fulfilled by AutoCAD Mechanical. The overall rating of AutoCAD in terms of fulfilment of the educational requirements is 86.3%.

A relatively low rate of meeting the requirements is observed in the construction of solid features (2 of 5), threaded fasteners (1 of 2), introduction to solid modelling (17 of 22), working drawings and assemblies (8 of 9).

TABLE 7. Assessment of AutoCAD Mechanical in terms of meeting CAD II requirements.

Number	Content	No of requirements	No of fulfilled requirements	Percent of fulfillments
1	Introduction to solid modelling	22	17	77.3
2	Construction of solid features	5	2	40.0
3	Tolerances, surface quality marks	7	7	100
4	Geometric dimensioning and tolerancing	8	8	100
5	Working drawing and assemblies	9	8	88.9
6	Threaded fasteners	2	1	50.0
7	Non-threaded fasteners, springs	3	3	100
8	Mechanisms	7	7	100
9	Welding	1	1	100
10	Descriptive geometry	9	9	100
<b>Overall</b>		<b>73</b>	<b>63</b>	<b>86.3</b>

## 4. Conclusions

AutoCAD Mechanical has been assessed in terms of the fulfillment of requirements of two main technical drawing courses, which are available in the mechanical engineering curriculum. This assessment is not meant as a benchmarking but checks whether AutoCAD Mechanical meets the educational requirements.

It is concluded that AutoCAD Mechanical meets most of the requirements (96 of 104) of the CAD I course, which focusses on part drawing. Relatively weak points of AutoCAD are in multiview - 3D model associativity and basics of assembly drawing. That is, AutoCAD Mechanical can be effectively used in the CAD I course education.

AutoCAD Mechanical meets 63 of the 73 requirements of CAD II, which focusses on assembly drawing. It is relatively weak in construction of solid features, 3-D threaded fasteners and the introduction to solid modelling. AutoCAD Mechanical is good in both courses, but is not as good in CAD II as in CAD I.

The overall rating of AutoCAD in terms of the fulfilment of educational requirements is 92.3% and 86.3% for CAD I and CAD II, respectively. The slightly lower percentage of 86.3% for CAD II is due to the fact that parts, fasteners (threaded and non-threaded), mechanisms, gears, cams, bearings, etc. occupy an important portion of CAD II which focusses on assembly drawing. Although, creating parts and fasteners of the assembly is possible with the knowledge of CAD I, assembling the parts or drawing the parts parametrically, which is very important for assemblies, is not as easy with AutoCAD as with a solid modeling software. Moreover, some features are not possible with AutoCAD, as mentioned in Section 3.2.

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