DELIVERY OF EINGEERING PROCESSES SAFELY AS A TEAM

2D CAD drawings

NAME: MOHAMMED GHALIB AL-HOSNI
NUMBER: 1615
CLASS: 7A

Report aim

The following report discusses the usage of a CAD software in drawing two separate engineering drawings, one being an electrical circuit with the other being a mechanical drawing, with the electrical circuit being an Astable Multivibrator while the mechanical drawing being a flange coupling. The aim of the discussion is to highlight the tools and commands that were utilized during the drawing process and also showcase the drawing methodology.

Contents table

CAD software utilized:

Utilized tools

Brief

Due to the use of a CAD drawing software for drawing the required engineering drawings, a variety drawing tools were utilized in order to create the required features and shapes while certain computer aided commands were utilized in order to fully utilize the CAD drawing process by making overall drawing procedure more convenient and easier. As such, the following section will be discussing the tools and commands that were used during the drawing process of both the electrical and mechanical drawings.

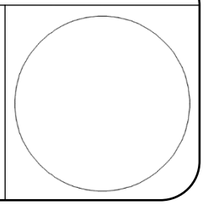
Drawing tools

Line

The basic drawing line is simply used to create a linear line and is undoubtedly the most utilized drawing-based tool throughout both of the given drawings as not only is it used to draw continuous lines, but rather it was also used for drawing hidden and centerlines. In addition, both drawings have a lot of straight shapes, and as such around 85% of the drawing is simply made up of simple line commands.

Circle

The circle drawing tool is used to draw circles with a variety of drawing options such as 2 or 3 point and Tangent based drawings. Dimensions for the function were given in radius as it is the more standard approach and although the stated tool was mostly used for drawing simple circles, it was also utilized in making arcs in places where it was easier to do so then utilizing the specific arc tool

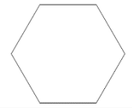


Rectangle

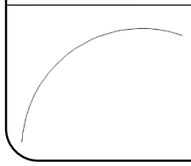
Although rectangular and square shapes could be easily drawn with the use of the line tool, it is more convenient to do so with the use of the rectangle drawing tool as it provides the user with an easier access to the required height and width.

Polygon

The polygon tool initially prompts the user to choose the number of sides, after which, the user picks a point and provides a radius for the polygon. Although the tool was not used extensively, it was extremely convenient when used.



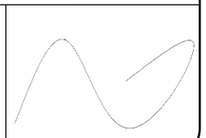
3-point arc



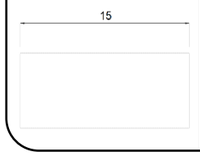
With the use of the 3-point arc tool the user is required to choose three different point in which an arc is going to revolve around. The tool was extremely important when drawing any curved sections seen throughout either drawing. Additionally, it was useful when providing some details especially to the mechanical drawing.

Spline fit

The spline fit tool provides the user with a curved object with a constantly altering radius, which can produce a string line shape that is generally utilized for drawing the side view of a non-uniform shape, which was required to be done in one of the given drawings.



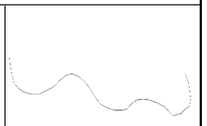
Dimensions



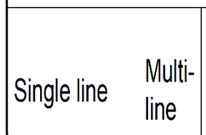
Dimensioning is an essential part of technical drawing, and in the given mechanical drawing, both linear and radius dimensions were added with the use of the dimensioning tool provided in the drawing software. With the dimensions text being placed over the line and always being parallel to the line it sits on.

Sketch

The sketch drawing tool permits the cursor to move freely and not be restricted while also placing lines onto the drawing board. Due to the line not being accurate, the tool was not frequently used, however, it when it was utilized, it was used for visual representation of a line that required a hand-drawn approach.



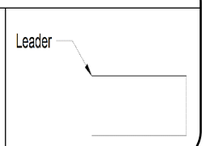
Multiline/Single line text



The text tool simply allows the user to input text onto the drawing board. Text was inputted onto the drawing in the form of both single and multiline, with each type being used according to the visual requirements and placement orientation of the text.

Leader

The leader is a tool allows that user to input an arrow onto the drawing, with the other end of the line containing a given object which by default is a text block, however, it can be switched to add a created block. The stated tool was in order to point at and represent what a specific section of the drawing exactly is.



Commands

Trim

The trim command is the one that was most utilized throughout the drawing process, what the command does is that it allows for the user to delete a section of a single object given that it has intersected with another object, therefore "trimming" the ends of the object which is extending through a secondary object.

Move

The move command simply allows the user to move a set of objects throughout the drawing board, the command itself was extremely convenient in ensuring absolute accuracy by allowing the user to move the object even for very small margins, which was useful in various location throughout both drawings.

Mirror

The mirror command allows the user to select a number of objects, after which the mirror command vertically flips them after a mirror line is selected, which is a line that states the center point from which the mirrored object is going to be placed in accordance to the original one. The mirror tool was utilized multiple types during the drawing process as it decreases the time needed and ensures symmetry and accuracy.

Scale

The scale command simultaneously decreases or increases the size of selected object after being provided by a set ratio from the user, with 1 being the default/current size of the drawing. The command was only used to fit the drawing into the drawing border by utilizing 0.5 as the chosen scale, which translates to 1:2. This was used for the mechanical drawing as it was initially too big for the required sheet size of A3 (297 x 420mm).

Explode

The explode command separates a single object into the individual components which are used to make it up. This command was mostly utilized when working with squares as it separates said them into the 4 lines that make them up, which convenient when trying to edit only a single line of the square that requires a specific linetype or lineweight while the rest do not necessitate any change.

Join

The join command is the opposite to that of explode, in that it connects a group of selected object and merges them into one given that they were initially interconnected. Although the use of this command was not necessary, it was useful in making the drawing neater and easier to work with by connecting lines of the same type so that the entire line can be selected in one click rather than having to select each individual piece of the line

Hatch

Hatching allows the user to shade a closed object by selecting its internal area, the commands also provides the user with a variety of hatch patterns to select from as to choose the one most adequate for what the shaded area represents. This saves a lot of time as it entirely removes the need to manually shade an object.

Offset

Offsetting an object provides the user with an identical outline of the selected object that either inscribe or circumscribe the original one. This command was mainly used when creating the border for both drawings as to create the internal border that is inscribed within the border representing the actual size of the page.

Demicontinue

Demicontinue works alongside the dimensioning drawing tools, what it does is that it allows for the user to input an additional dimensioning line that extends from one that was already drawn to form a chain dimensioning arrangement.

Center mark

Center mark is an annotation tool used to automatically input a centerline on a circle, the command that was not frequently used although it was convenient as the user would not have to manually draw the centerline.

Block

Blocks are a method to create a custom object by selecting a manual drawn shape which contains a variety of default object which can then be joined together using the block creation command and utilized as any other drawing tool.

Cursor snapping

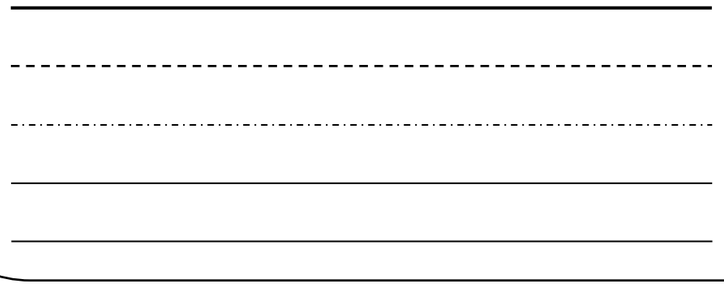
The cursor snapping command causes the user's cursor to snap onto various section of an object as to ensure accuracy and that the starting or ending position of a line is exactly on the section the cursor has snapped on. Although there are a variety of object sections that could be made detectable by cursor snapping, the ones that were chosen were endpoints, midpoints, circle centers, and intersections.

Cursor restriction

Cursor restriction commands simply restricts the user from moving an object in a specific manner, with orthogonal restriction preventing diagonal movement and angular restriction preventing the user from freely moving their cursor whenever they are at a right angle, this allows for more precise drawing as the drawn line or object is ensured to be straight and not ever so slightly tilted.

Layers

Layers is a prominent feature seen in CAD drawing applications as it allows for the user to separate lines into different groups or "layers", with each layer having its own set of properties that can be set by the user. Layers were extensively used for setting the lineweight and linetypes of the drawing, as it was especially convenient since any changes that were required to a specific line group (e.g: all hidden lines) could be done through the layer, which automatically edits the properties of every line in the group rather than having to manually edit the property for each line that required said change. The layers that were specifically added to the drawings were ***Continuous****, which had continuous lines of lineweight 0.7,* ***Hidden****, which contained dashed lines of lineweight 0.5,* ***Centerline****, containing centerlines with a lineweight of 0.35,* ***Dimensions****, which contained continuous and a lineweight of 0.35, and finally,* ***Details****, which is an additional layer for details and markings, which also had continuous lines and a lineweight of 0.35.*

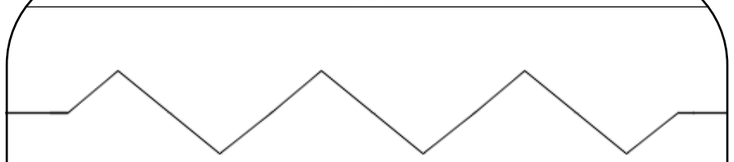


Electrical drawing

Astable multivibrator

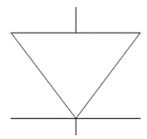
The shown electrical circuit represents a 555 timer, an electronics component that is utilized in a variety of ways such as generating pulses over a set frequency or to control a required time delay. The specific configuration shown on the drawing is of an Astable 555 timer, a circuit configuration that can be used in order to generate square signals, and therefore is able to provide Pulse Width Modulation, a type of signal that emits HIGH and LOW signals over a set frequency as to emulating an analog signal while still outputting digital ones, additionally it can also be used in oscillation and as a primary frequency source. The main features that can be seen on the circuit are the ***555 timer, NAND logic gate, diode, resistor, capacitor, control signal, power supply inputs, ground, wire connection points,*** *and the* ***current flow arrow.***

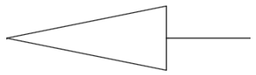
Resistor



Drawing the resistor conventionally would have constituted drawing diagonal lines of 45-degree angle and repeating the process 3 times, which would have been an extremely inconvenient procedure. As such, the method that was utilized was to utilize a zigzag linetype that was loaded from the AutoCAD's linetype library to then simply draw the zigzag line as if it were a normal continuous line. However, the distance and overall height of the zigzag lines were too small initially, and therefore the linetype scale was increased until it was deemed fit for the drawing.

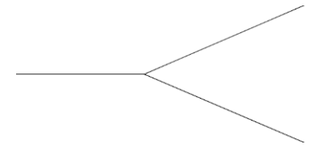
Diode and current flow arrow





The diode was initially drawn by inserting a rectangle possessing the required length of the diode's triangle. After which, a line was drawn from two adjacent corners of the rectangle to the opposite side's middle point as to form the triangle within the original rectangle. After which, the side lines of the rectangle were trimmed off as to leave the line showing the bottom of the triangle and the line representing the negative side of the diode. The same procedure was used in drawing the current flow arrow, with the difference being that the initial rectangle was longer and the line used to showcase the negative has also been trimmed off.

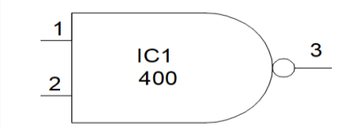
Control signal



The control signal feature was drawn by initially inserting a straight line of the required size and then giving it an angle of 45-degrees, with the angle being calculated with the use of the angular restriction command as in addition to snapping the line at right angles, it also provides an angle parameter that can be seen and edited by the user. After which, the same process was done for drawing the adjacent line, with the only difference lying in the angle that was inputted, in that for the second line -45 degrees (additionally 315 degrees could have also been used) was utilized instead of 45-degrees.

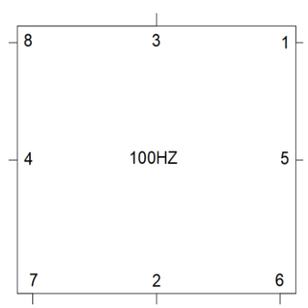
NAND gate

The NAND gate was initially drawn as a simple rectangle, after which a circle of the same diameter as the width of the rectangle was drawn from the middle point of the rectangle's right side. After which, the lines remaining within the area of the rectangle and circle were trimmed off. In order to add the smaller circle differentiating between NAND and AND gates, the cursor was snapped onto the middle point of the original circle's right side, from there, a circle the third of the initial one's size was drawn and then later moved to make it so the side of the circle locks onto the rightmost tip of the bigger circle. After which any remaining lines were trimmed off.



555 timer

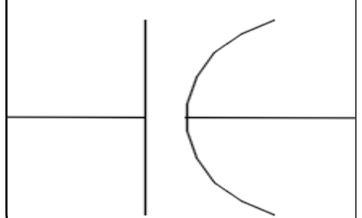
To draw the 555 timer, the rectangle drawing tool was used to create the primary shape, after which, in order to add the lines extending from the component, around a tenth of the total distance was used as a distance between the edge and the lines drawn for pins 1, 6, 7, and 8, while the pins located in the middle were simply drawn by turning on cursor snapping to the middle point of the line, which helped in achieving both precise measurements and convenience.



Ground

The ground symbol was firstly drawn from longest line, and after which adding two identical lines underneath by offsetting it. Following that, the second line was shortened by 2.5mm from each end while the third one was shortened by 5mm.

Capacitor



The straight side of the capacitor was merely drawn as a straight line. However, in terms of the curved side of the capacitor's symbol, it was initially drawn as a circle that was then trimmed off by drawing a straight line through the middle and trimming the other side, after which, a slight trimming of the remaining semicircle from its tips was additionally done in order to make the drawn symbol look more than the actual one as the curved side is not a perfect semicircle.

Mechanical drawing

Flange Coupling

The shown mechanical drawing is one of a flange coupling, a device that is used in order to connect two different rotating shaft together in order to transmit the kinetic power from the driving shaft – the shaft that is initially rotating due to an engine or other moving mechanisms – to the driven shaft, which is the shaft that starts rotating due to the driving shaft's rotation. This process is done by connecting the two shafts together with the use of a flange, a component that stands between the two shafts in order to lock them together with the use of keys and nuts and bolts. Overall, the main component that make up a flange coupling are the base, also known as the ***Flange****, the* ***Hub****, which is the section attached to the flange, the two* ***Shafts*** *which were just stated,* ***Keys*** *to lock the mechanism and prevent it from rotating, and finally the* ***Nuts & Bolts*** *which are used to hold together the entire device.*

Front view drawing methodology

Flange

The flange was the first component that was placed on the drawing as it provided the placement and positions that each of the following drawings were going to require. The flange itself was drawn was by simply drawing a rectangle, and after doing so simply adding a line that passes through its midpoint.

Hub

The hub was drawn by simply adding a rectangle onto the flange that is around 45% of the flange's size in length. After which said rectangle was mirrored from the middle point of the flange's width so that it is also added onto the other side of the flange as well.

Shaft

The shaft was initially drawn as rectangle connected to the Hub which is also around 45r% of the Hub's size in length. After which, the spline fit drawing tool was utilized in order to create the shape seen at representing the end of the shaft as to follow the conventional drawing method of how a shaft is represented. The way the required tip shape was acquired was by starting the spline fit tool at the bottom outer facing corner of the shaft, after which choosing the midpoint, moving the cursor to the right, choosing the top corner point, and then going back to the midpoint, which results in the thread-like shape which is required. Finally, hidden lines were added from the corner of the shaft and extending all the way towards the middle of the flange. After the stated process has concluded, the shaft was mirrored onto the other side of the front view drawing.

Nuts and bolts

Although drawing nuts and bolts might seem like a relatively simple task considering their rectangular shape they might be assumed to have when seen from the side, however, the conventional method of drawing them is to add details to their ends to represent the curvature. As such, the following process was gone through when drawing the nuts and bolts' side view:

Initially, two rectangles of equal dimensions were drawn on either end of the flange, with the one on the left showcasing the bolt's head, while the one on the right representing the nut. Additionally, the nut also has an additional rectangle drawn to its right from the middle that is around 75% its size which represents the end of the bolt. Following this, a line that is 12.5% of the bolt head's width was drawn on top of the bolt head, after which, a straight line was drawn through the entirety of the head's area. From there, a 3-point arc was drawn from the initial intersection point of the line with the bolt head, to the quarter point of the bolt head's length, and finally to the midpoint of the drawn line, the same procedure was then done but form the bottom of the bolt head's length rather than from the top, after which, a line was drawn from the end tips of the arc to the rightmost midpoint of the bolt head. The entirety of the stated process was done in order to represent the bolt head's curvature and its current orientation. Finally, the initial line drawn for the intersection was trimmed, and the newly drawn lines were mirrored so that they also appear on the nut, however, the nut included said lines on both ends rather than its front only.

In addition to the bolt's head and nut, the tip of the bolt also requires to be drawn in a specific manner to show an idea of how it might look in real life, which was done by offsetting both ends of the initially drawn square for the tip by a small margin of around 5% the original length, from there, the tips of the rightmost corner squares are to be connected with a line as to create a triangle, any unnecessary lines are trimmed off.

Keys

To begin with, simply the visible ends of the keys were drawn onto the shaft by adding two rectangles onto it, one being on top of the shaft at the left side of the drawing, with the other being on the shaft's side which is one the right side of the front view drawing, with both keys extending from the hub by the same distance. After which, hidden lines were added from the tips of the keys going all the way to the middle of the flange. In terms of the key at the top of the shaft, it was necessary to represent the depth of the key since from the front view's perspective the key's side is showcased, this was done by adding a hidden line from the bottom left corner of the key that is 15% longer in length that the visible section at the top, after which, another hidden line going from the one that was just created goes is extended until the middle of the flange.

In terms of the drawn detail for the key, it was added in order to represent the groove in a more conventional manner as well as showcase and add dimension for the groove's depth. This was done by drawing an enlarged version of the original key by a scale of 2:1 without showcasing any of the hidden lines. After which, the line representing the hub as added and a circle was placed around the stated drawing, with any bits of the drawings extending from the circle being trimmed off. After which, a short line that is 35% of the key's width until the hub was extended from the left side of the key, which was placed at the same distance from the top of the key to the start of the groove. Thenceforth, a hand-drawn line was added from the extended line's end with the use of the sketch tool and was drawn going under the key and until it exited the circle. From there, any of the sketched line that went out of the circle was trimmed off. The reason that only one detail drawing was added was due to the other key having the same exact dimensions, and therefore, the detail drawing that was added could be used for either key.

Side view drawing methodology

Flange

The flange was the first circle to be placed and was simply drawn using the circle tool, and thenceforth was used as the midpoint for each of the following features to be drawn.

Hub

Just like the flange, the hub was drawn by simply using the circle tool. It is placed within the flange's circle and is around 45% of its size in diameter.

Shaft

The shaft was dawn with the use of the circle with the shading seen within its area being added using the hatch command. It is placed inside the flange's circle and is 20% of its size in diameter.

Nuts and bolts

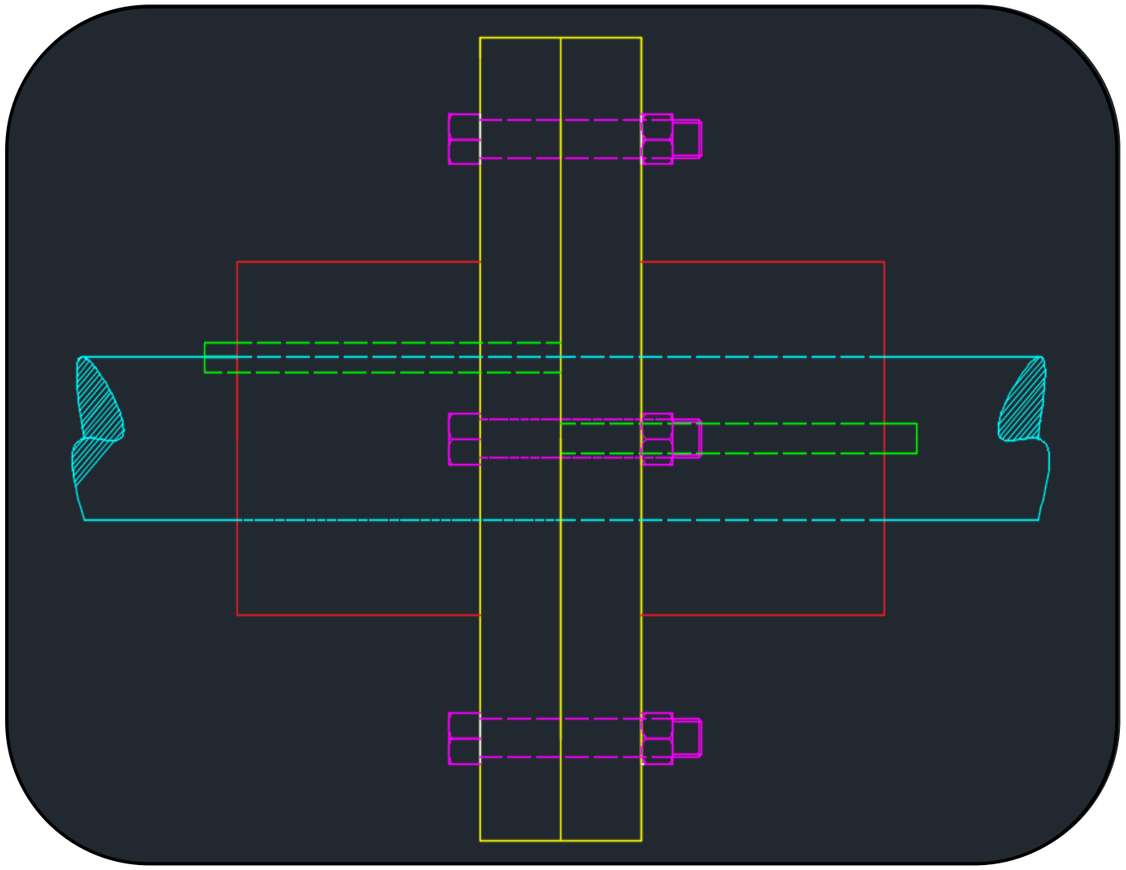
To draw the nuts and bolts, initially, a centerline circle was added that is around 75% of the flange's diameter. After which, horizontal and vertical lines were extended from the center point of the flange's circle going towards each side which was done in order to create an intersection between the drawn centerline as to show the midpoint of each side. After which, a small circle was drawn from said intersection to represent the thread, which was also drawn in hidden lines as it would not be visible from the left side of the flange coupling. Thenceforth, the polygon drawing tool was using to draw a hexagon which was 30% bigger than the thread's circle in diameter, with its midpoint also extending from the created intersection. Additionally, another circle of the same diameter as the hexagon was drawn from the intersection to draw the front view of the bolt in the conventional manner.

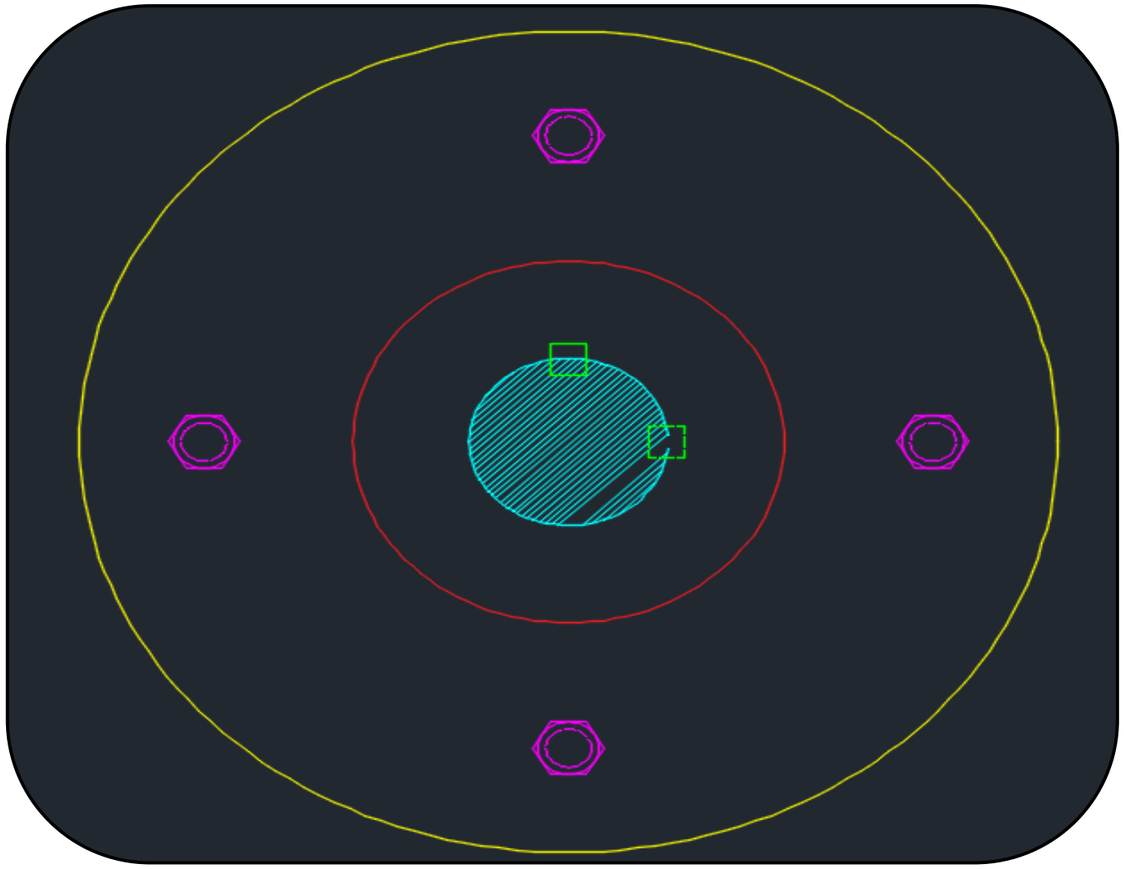
Keys

The keys were initially drawn by adding the squares with the required dimensions on top of the shaft while making sure the section within the shaft on the front view is also represented on the side view as to show the keys are within a groove. In terms of the the top key, the section within the shaft was trimmed off and replaced by hidden lines to showcase the key is indeed within the shaft. However, with regards to the side view, the entirety of the square's line type was changed to hidden lines as said key would not be visible from the left side of the flange coupling, which is the side that is shown in the drawing due to the first angle projection that is utilized. Additionally, due to both keys possessing the same dimensions, they were also both given the same groove depth, and therefore the depth of the top key is the same as the one seen for the side key.

All features drawn on the side view were ensured to be of the same size and alignment to in comparison their equivalent in the front view with both of which being placed on the same height in accordance to the drawing board's placement.

2D CAD drawings





Flange | Hub | Shaft | Nut-bolt | Keys

"*A display connected to a digital computer gives us a chance to gain familiarity with concepts not realizable in the physical world. It is a looking glass into a mathematical wonderland.*"

-Ivan Sutherland

Developer of the first CAD software