1. The task:

- Create this mechanical drawing in AutoCAD system.
2. The solution steps

Start the AutoCAD system.

To create a new layer, click on in the Object Properties toolbar, the Layer & Linetype Properties dialogue box, illustrated below, appears.
The Layer tab is always selected by default. Now click on the "New" button. A new layer called "Layer1" is automatically created in the layer list below layer 0. As you can see from the illustration, the layer name is automatically highlighted for you so that you can give the layer a more meaningful name. When you have entered an appropriate name, press the key to complete the operation. You have now created a new layer and given it a name. Notice that by default it has been assigned the colour white and the linetype "Continuous".

Click on the colour icon in the layer list associated with the layer you want. Notice that all layers have their own colour icon and that this changes to display the layer colour. Clicking on the icon brings up the Select Color dialogue box, shown on the right. You can select any of the 255 standard AutoCAD colours by picking on the colour palette or by entering the colour name or number in the text edit box. When you have selected the colour you want, click on the "OK" button to set the colour. AutoCAD uses only 255 colours plus the drawing background colour, irrespective of the capabilities of your video display.

In the same way that you can assign a colour to a layer you can also assign a linetype to a layer. Click on the current linetype name associated with your layer in the layer list. By default, layers have the "Continuous" linetype. Clicking on the linetype name brings up the Select Linetype dialogue box, shown on the right. You will notice that the "Continuous" linetype is the only one listed. That's because all linetypes, except "Continuous", are stored in an external file and have to be loaded before they can be used.

In the same way that you can assign a linetype to a layer you can also assign a Lineweight to a layer.

We use this layers:

- Thick line, colour: green, linetype: Continuous, lineweight: 0,60 mm;
- Thin line, colour: yellow, linetype: Continuous, lineweight: 0,15 mm;
- Axis line, colour: purple, linetype: Dashed, lineweight: 0,15 mm;

Using the Line command, a line can be drawn between any two points picked within the drawing area. Lines are usually the first objects you will want to draw when starting a new drawing because they can be used as "construction lines" upon which the rest of your drawing
will be based. Never forget that creating drawings with AutoCAD is not so dissimilar from creating drawings on a drawing board. Many of the basic drawing methods are the same. Draw a rectangle (210 mm x 297 mm) in the Thin layer with Line command.

The Offset command creates a new object parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen by the user with a pick point. You can offset lines, arcs, circles, ellipses, 2D polylines, xlines, rays and planar splines. Use this command with 5mm value from the yellow frame. Create this in Thick layer.

Create 5 parallel lines. See below. This is the sketch of the titleblock.
The Trim command can be used to trim a part of an object. In order to trim an object you must draw a second object which forms the "cutting edge". Cutting edges can be lines, xlines, rays, polylines, circles, arcs or ellipses. Use this command and create the exact titleblok then use the Text command and create the text in the titleblock.

Start the drawing in the prepared frame. First draw the middle of V-pulley with Line command.
Next step, modify the lines intersection points with Chamfer command. The Chamfer command enables you to create a chamfer between any two non-parallel lines as in the illustration below or any two adjacent polyline segments. Usually, the Chamfer command is used to set the chamfer distances before drawing the chamfer.

Continue the drawing. Draw the contour of the rest then use the Fillet command. The Fillet command is a very useful tool which allows you to draw an arc between two intersecting lines or adjacent polyline segments. You first need to use the command to set the required radius and then a second time to select the two lines.
Draw the V profiles and fillet.

Now draw the bottom side of section view. Use Line, Offset Trim, Chamfer and Fillet commands.
Now draw the half side view of pulley. Half side view is good here, because this part is a symmetric part. Draw some Circles with Circle command. The Circle command is used to draw circles. There are a number of ways you can define the circle. The default method is to pick the centre point and then to either pick a second point on the circumference of the circle or enter the circle radius at the keyboard. Trim the unnecessary sections.
Draw the holes.
Create the details views. Use the Copy command.
The Copy command can be used to create one or more duplicates of any drawing object or objects which you have previously created. Copy is a very useful and time-saving command because you can create very complex drawing elements and then simply copy them as many times as you like. After the copy, use the Scale command.
The Scale command can be used to change the size of an object or group of objects. You are prompted for a pick point about which the selection set will be scaled. Scaling can then be completed by picking a second point (not always easy because it can sometimes be difficult to precisely control the scaling) or by entering a scale factor at the keyboard.

Draw B-B section’s arrows with Fast Section Line command.

This section describes the options and commands available for dimensioning drawings and how to use them. The correct use of AutoCAD’s dimension tools is the key to producing clear and concise measured drawings. If you just need to quickly find a description of the various dimension commands, click on the appropriate button on the QuickFind toolbar below.

When you create dimensions, AutoCAD automatically creates a new layer called “Defpoints”. This is a special layer which cannot be deleted or renamed. AutoCAD uses this layer to store dimension information and you can effectively ignore it.

The Linear Dimension Command. As the name suggests the Linear dimension commands are used to dimension along straight lines. You can use this command to generate horizontal and vertical dimensions.
Creating a linear dimension is easy. All you have to do is start the command, specify the two points between which you want the dimension to be drawn and pick a point to fix the position of the dimension line.

The Radius Dimension Command. The Radius command is identical to the Diameter command except that the dimension measurement is a radius rather than a dimension and the resulting dimension text is prefixed with a "R" to indicate radius.

The Diameter Dimension Command. You can use the Diameter command to annotate a circle or an arc with a diameter dimension. To achieve this simply start the command, pick a point on the circumference of the circle, pick a second point to determine the length of the leader and then add the dimension text or Return to accept the default.

The Dimension Style Command. The Dimension Style command can be used to change the appearance of dimensions. The best method is to create a new style before you start creating dimensions so that you can leave the STANDARD style as a default option. Having created a new style from STANDARD you can then apply any modifications you generally require to the parent style and then more specific modifications to the child styles in order to create a style family.

Dimension styles are created using the Dimension Styles dialogue box. The dialogue box is shown on the right. As you can see from the dialogue box, a style is applied to a family of
dimensions. By default, any style changes are made to the parent. Each style parent has six child styles. The child styles, Linear, Radial, Angular, Diameter, Ordinate and Leader can be used to modify the parent style when that particular type of dimension is used. For example, you may like to use a tick rather than an arrow head for your dimensions but this isn't really appropriate for a leader, so the Leader child style can be changed so that leaders will always be drawn with an arrow head whilst all other dimensions of the same style family are drawn using ticks.

The hatch command can be found on the draw toolbar, or the draw drop down menu. When entered, a dialogue box appears as shown below. Hatching is used to add shaded patterns to objects and shapes within an Autocad drawing. Hatch patterns can be used to indicate a material to be used, such as a concrete hatch. Alternatively it could be used to make an area of a drawing stand out. The above hatch was created by using the pick point method. The hatch stays within all the solid lines.
Your finished drawing should look like the one below.