

**Northern Technical University**

**Technical Engineering college/Mosul**

**Department : Technical engineering of power mechanics**

**Third Level**

**Subject: Computer Application 3rd level**

**Autodesk Mechanical Desktop 2009 (MDT)**

**Lecturer**

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## **Autodesk Mechanical Desktop 2009 (MDT)**

AutoCAD Mechanical software extends the capabilities of AutoCAD so that you can quickly create and manage mechanical drawings.

Autodesk Mechanical Desktop 2009 (MDT) software includes all the functionality of AutoCAD, plus libraries of standards-based parts and tools to help accelerate mechanical CAD.

**The objective** of using Mechanical Desktop design software package is to insert 2D or 3D mechanical parts (screws, washers, nuts, shafts, keys, bearings, springs, ...etc.). This software is used in mechanical design analysis and calculations were made after applying forces on the mechanical parts and inserting the data needed to perform the calculations.

# Screws

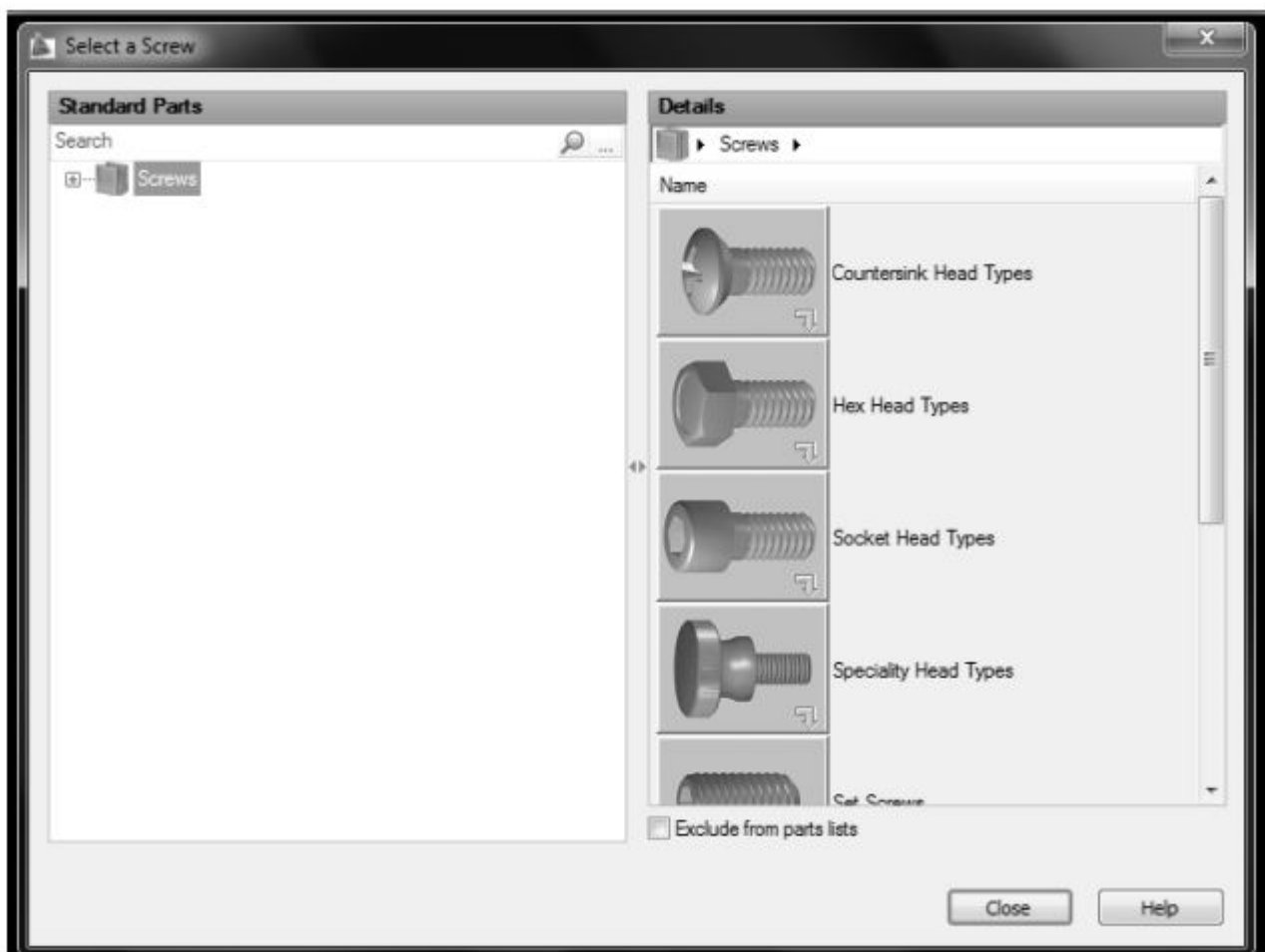
## How to draw a 2D or 3D Screw by using Mechanical Desktop?

1- Menu ⇒ content2D or 3D ⇒ Fasteners ⇒ Screws

2- In the command line write **AMSCREW2D** or **AMSCREW3D**

After a Screw dialog box appear, select a type of screw and view to insert.

Depending on the selected view there are differences in the insertion process.



## Command line options for 2D Screw

- *Front View/Side View*

Specify insertion point: *Click a point to specify an insertion point.*

Specify rotation angle <0>: *Specify the rotation angle using the pointing device, or enter a value.*

In the Nominal Diameter dialog box, select the size.

Drag size: *Specify the length using the pointing device, or enter a value.*

- **Top View**

Specify insertion point: *Click a point to specify an insertion point.*

Specify rotation angle <0>: *Specify the rotation angle using the pointing device, or enter a value.*

In the Nominal Diameter dialog box, select the size.

In the Length Selection dialog box, select a length.

## **Command line options for 3D Screw**

Select first point [Concentric/cylinder/two Edges]: *Select a first point or select the option you want to use to insert the 3D part.*

Select second point [Concentric/cylinder/two Edges]: *Select a second point. A dialog box named for the standard you selected is displayed in which you can select the nominal diameter for the standard part.*

Drag Size [Dialog]: *Drag the size by moving the cursor in insertion direction. If you select Dialog, you choose the part from a table.*

## **Command line definitions**

- **Concentric**

Select circular or elliptical edge: *Click the edge of the area where you want to insert the standard part. (1).*

Enter an option [Next/Accept] <Accept>: *Press ENTER.*

# Washers

## How to draw a 2D or 3D Washer by using Mechanical Desktop

- 1- Menu  $\Rightarrow$  content2D or 3D  $\Rightarrow$  Fasteners  $\Rightarrow$  Washers
- 2- 2- In the command line write **AMWASHER2D** or **AMWASHER3D**

After a Washer dialog box appear, select a type of washer and view to insert. Depending on the selected view there are differences in the insertion process.

## Command line options for 2D Washer

*Front View/Top View*

Specify insertion point: *Click a point to specify an insertion point.*

Specify rotation angle <0>: *Specify the rotation angle using the pointing device, or enter a value.*

In the Nominal Diameter dialog box, select the diameter.

In the Select Part Size dialog box, select the size.

**Note:-** To draw a 3D Washer see the previous lecture of 3D Screw drawing steps.



# Nuts

## How to draw a 2D or 3D Nut by using Mechanical Desktop?

1- Menu  $\Rightarrow$  content2D or 3D  $\Rightarrow$  Fasteners  $\Rightarrow$  Nuts

2- In the command line write **AMNUT2D** or **AMNUT3D**

After a Nut dialog box appear, select a type of Nut and view to insert. Depending on the selected view there are differences in the insertion process.

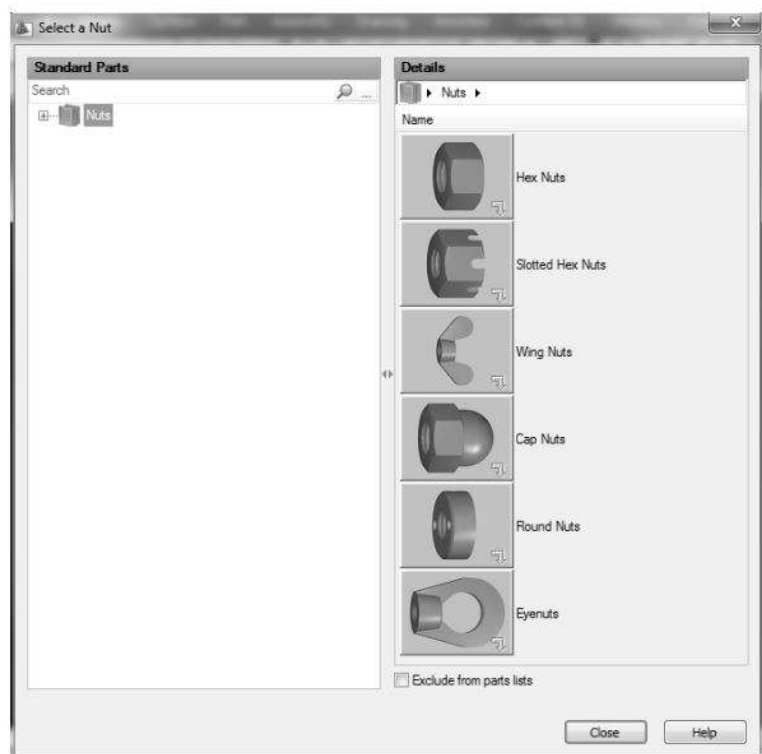
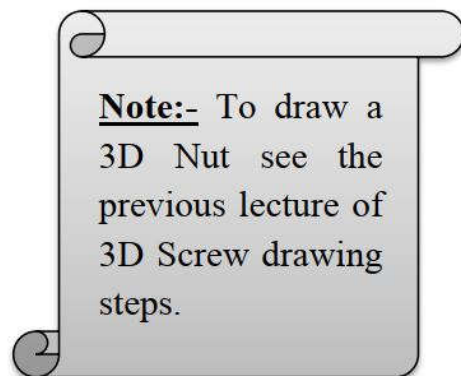
## Command line options for 2D Nut

*Front View/Side View/Top View*

*Specify insertion point: Select a point to specify an insertion point.*

*Specify rotation angle <0>: Specify the rotation angle using the pointing device, or enter a value.*

*In the Nominal Diameter dialog box, select the size.*

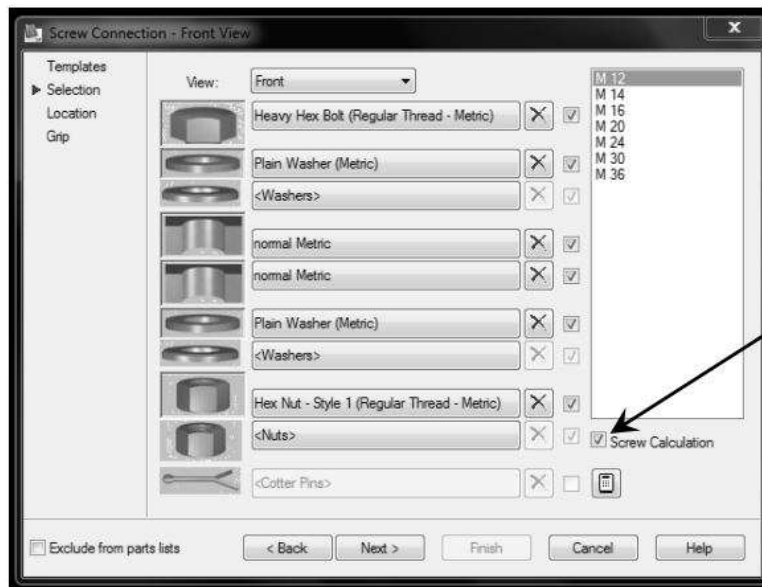


## Screw Connection

**Example 1:-** How to draw a 2D Sectional Front View/Screw Connection by using these parts:-

- a- **Screw:-** (M12) Heavy Hex Bolt Type (Regular Thread-Metric).
- b- **Washer:-** Two washer in each side (Plain washer (Metric)).
- c- **Holes:-** Two Through Cylindrical Holes/Normal Metric.
- d- **Nuts:-** (M12) Hex Nut - Style 1 (Regular Thread Metric).
- e- **Plates:-** GL1 and GL2 = 10.
- f- **Solution:-**

Firstly we choose Screw Connection parts from content 2D ⇒ Screw Connection ⇒ Now the Screw Connection dialog will appears as below :-

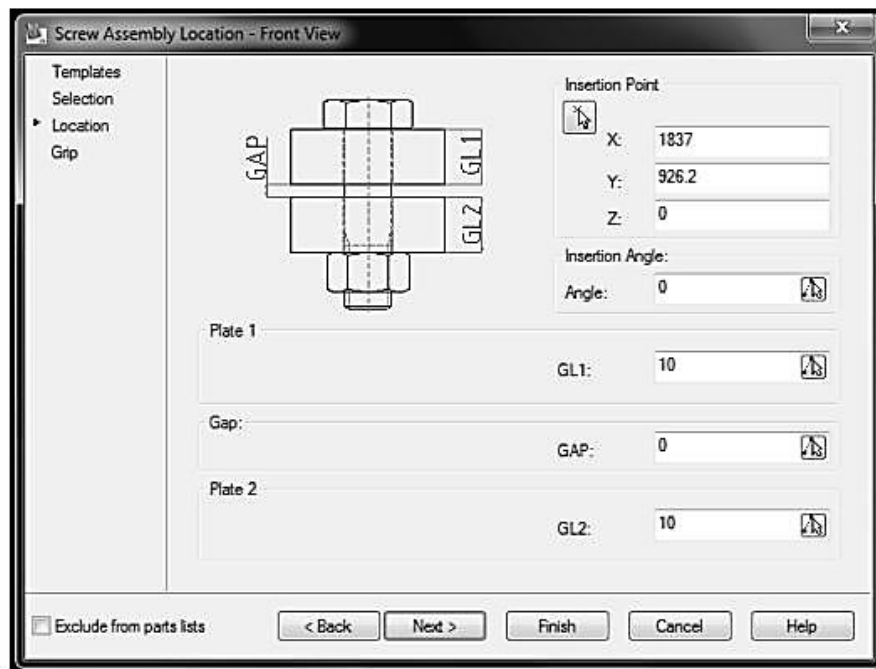


**Note:-** If you select this check box you can do a screw calculation also. (See Example 2).

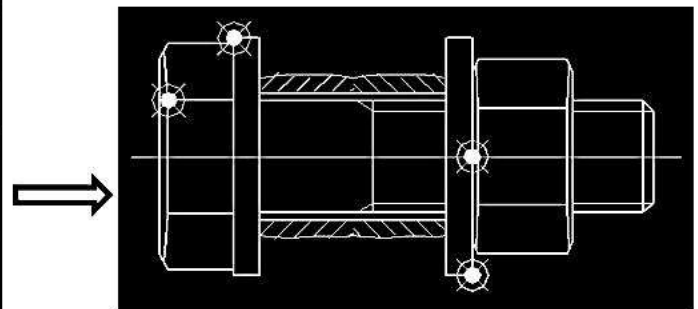
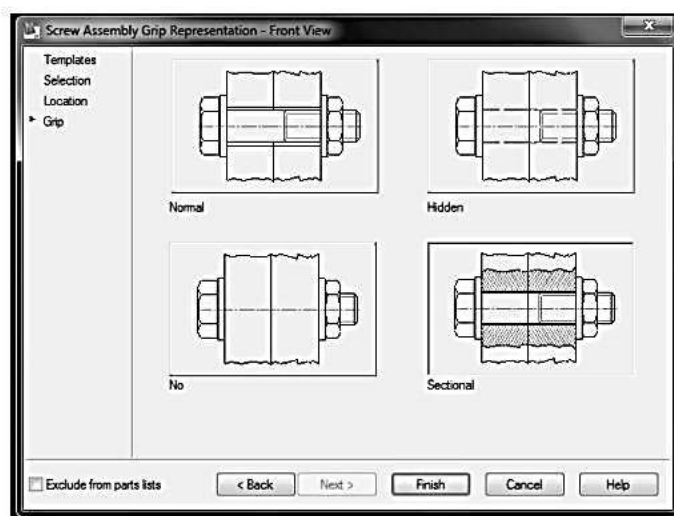
Then next and the program will ask in the command line to:-

- *Specify insertion point of first hole:* we will choose a point by using the mouse.
- *Specify endpoint of first hole:* we will choose a point by using the mouse.
- *Specify endpoint of second hole [Gap between holes]:* we will choose a point by using the mouse.

Then a dialog will appears as below:- from this dialog we should put the value of GL1 & GL2. Then next.



At this point we choose Sectional view from the dialog below and finish.



Finally the result

## Methods for Calculating Screws

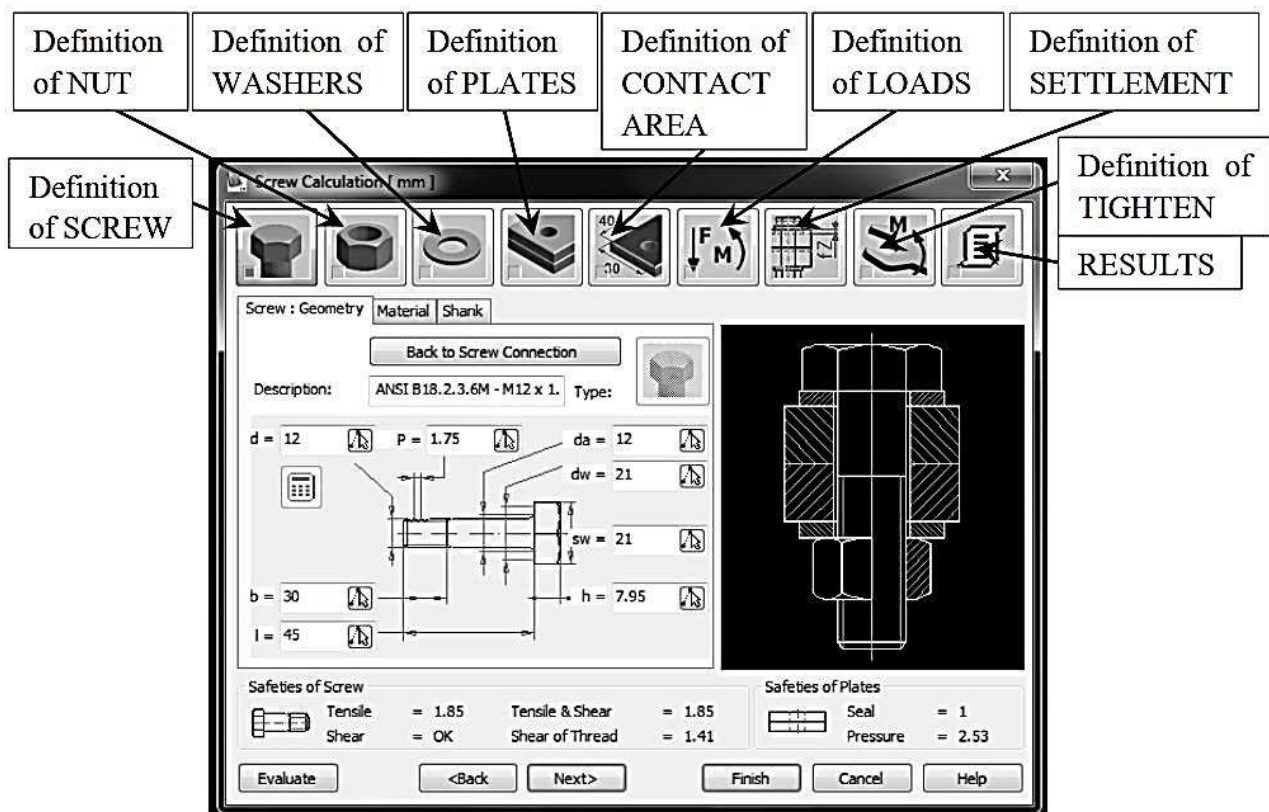
The Screw Calculation provides two different ways to calculate a screw connection:-

- Stand-alone calculation: All data and properties are specified by the user.
- Calculation of an existing screw connection: The user selects an existing screw connection to be calculated. All geometric and standard-related data is taken from the screw connection and cannot be edited.

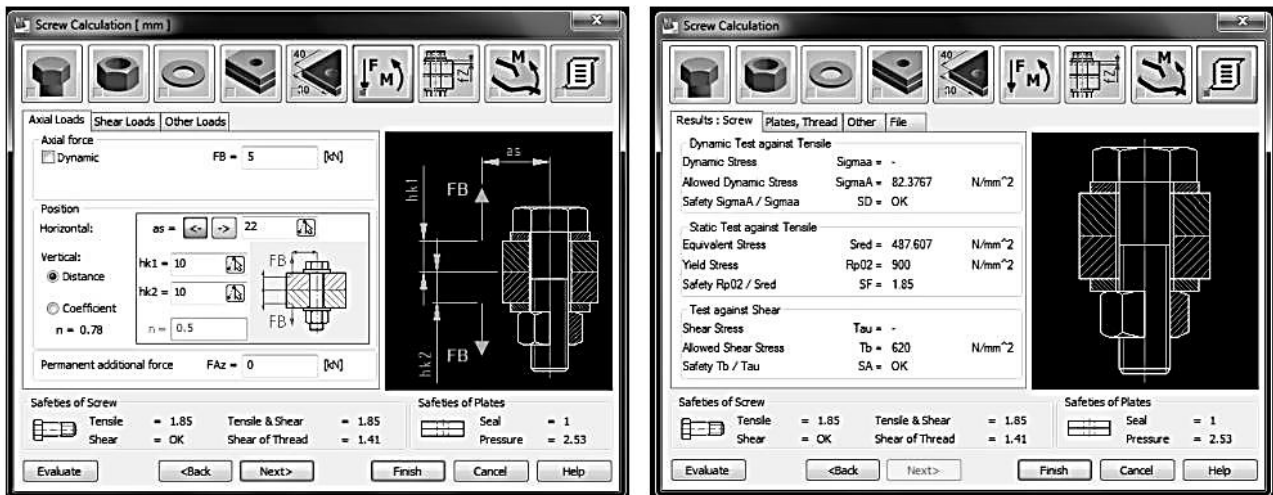
In this exercise, you use the stand-alone Screw Calculation. With the standalone calculation, you can calculate a screw connection without any prerequisites. You can specify the screw connection in detail (material, geometry, load, settlement and tightening properties). In this exercise, you are provided with the drawing of a screw calculation. Some values are selected from tables, some are entered manually, and some are taken directly from the drawing.

**Example 2:-** Do a screw calculation for Example 1.

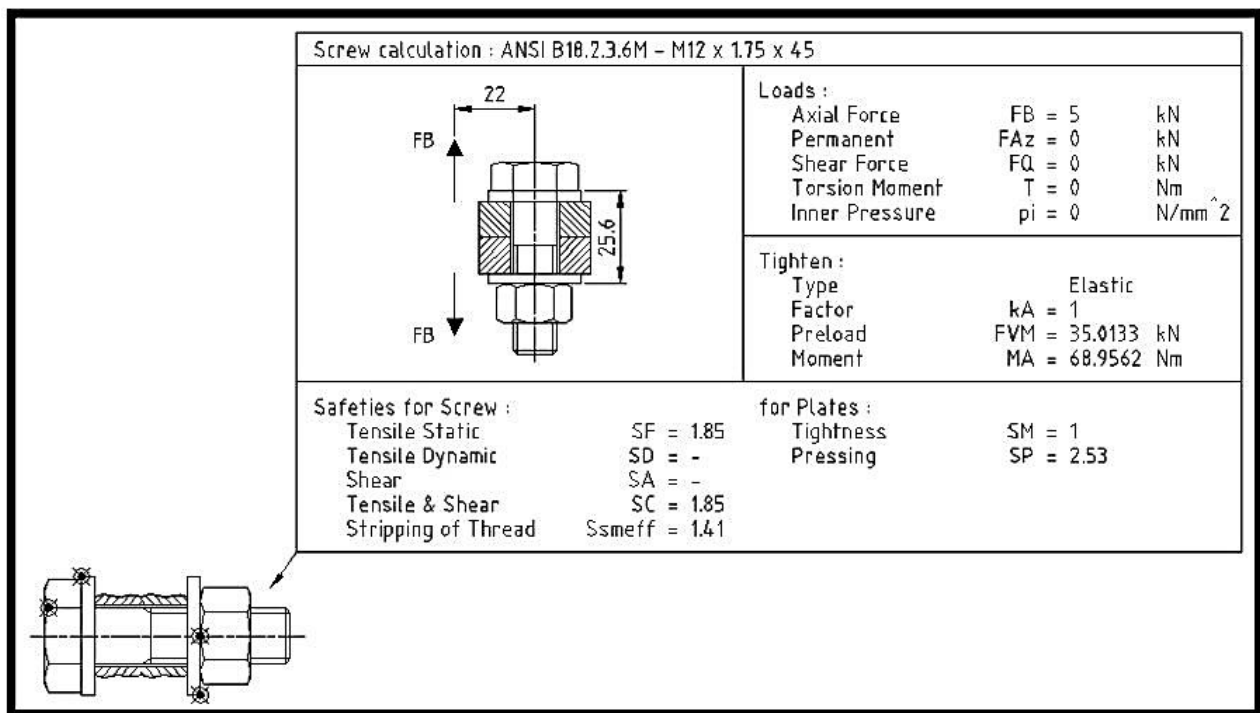
**Solution:-** If you select the check box for the screw calculation in the screw connection dialog box as mentioned in Example 1, then you can do a screw calculation. The figure below shows you how you can define a multiple choice (screw, nut, washers, plates, contact area, loads, settlement, tighten, and results) to define your design.



In the load icon you can put the force manually and then go to the results and see your design results.



Then click on finish to insert the data sheet as shown in the figure below



**Exercise:- Note** It is required that the ISO standard parts be installed for this tutorial exercise.

1- Open the file *tut\_std\_pts.dwg* in the Tutorial folder at:

- Windows 7 or Vista:

*C:\Users\Public\PublicDocuments\Autodesk\ACADM2009\Acadm\Tutorial*

- Windows XP: *C:\Documents and Settings\All Users\Shared Documents\Autodesk\ACADM 2009\Acadm\Tutorial*

The drawing contains a motor with a gearbox. Some construction lines are inserted to help you work through the tutorial exercise. The gearbox is not completed yet. We want to add standard components and show how easy it is to edit standard parts with an automatic update of the background objects. Before you proceed, you must enable mechanical structure. If you proceed without mechanical structure enabled, some command line prompts will differ from the prompts in the exercise.

2- Zoom in to the area of interest. On the command line, enter *ZOOM*.

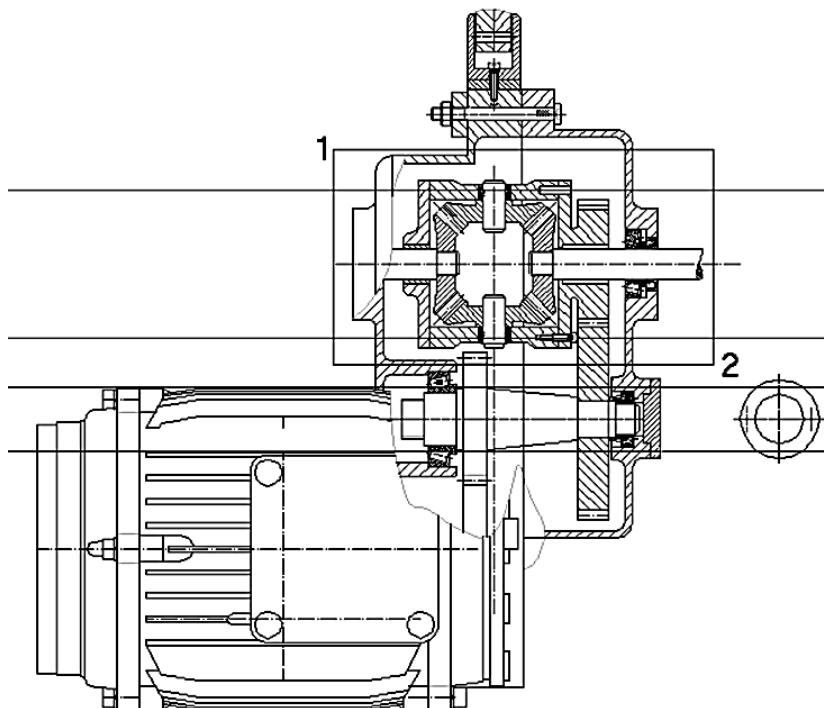
3- Respond to the prompts as follows:

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:

*Enter W, press ENTER*

Specify first corner: *Specify the first corner point (1)*

Specify opposite corner: *Specify the second corner point (2)*



4- Save your file under a different name or to a different directory to preserve the original tutorial file.

5- Insert a screw connection in the differential gear housing.

✓ To insert a screw connection

a) Start the Screw Connection command. On the command line, enter *AMSCREWCON2D*.

**b)** In the Screw Connection dialog box, click Screws and select Socket Head Types (ISO 4762 and Front View.) M4.

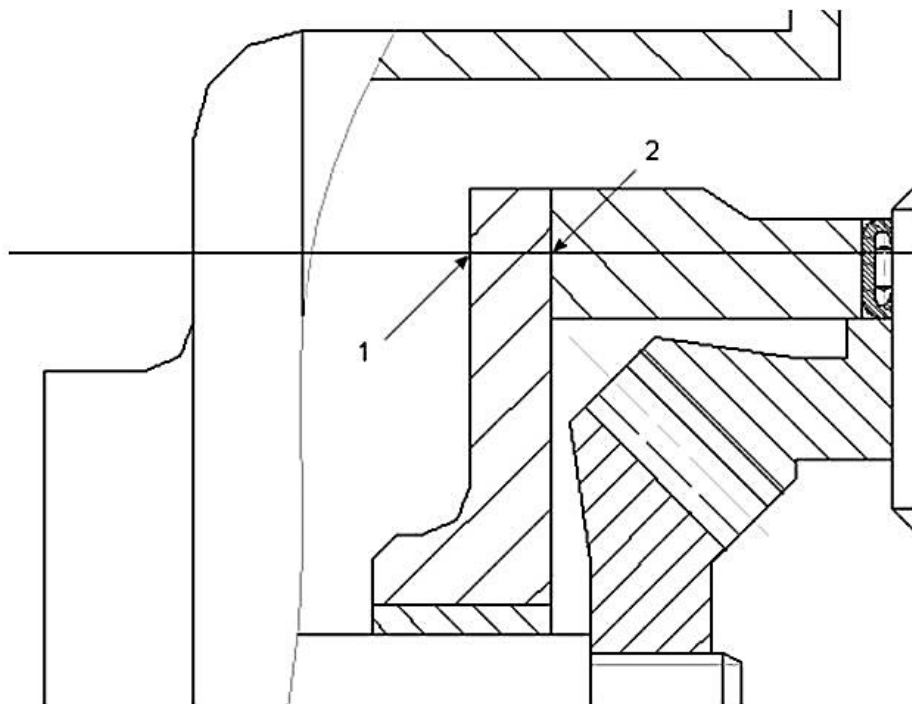
✓ In the Screw Connection - Front View dialog box, click the upper Holes button. Then select Through Cylindrical, and ISO 273 normal.

✓ In the Screw Connection - Front View dialog box, click the lower Holes button. Then select Tapped Holes, Blind, and ISO 262 (Regular Thread).

✓ **Respond to the prompts** as follows:

Specify insertion point of first hole: *Specify first point (1)*

Specify endpoint of first hole [Gap between holes]: *Specify second point(2)*



**6-** In the Screw Assembly Representation - Front View dialog box, click Next.

**7-** In the Screw Assembly Grip Representation - Front View dialog box, Choose Normal and click Finish.

**8-** Respond to the prompts as follows:

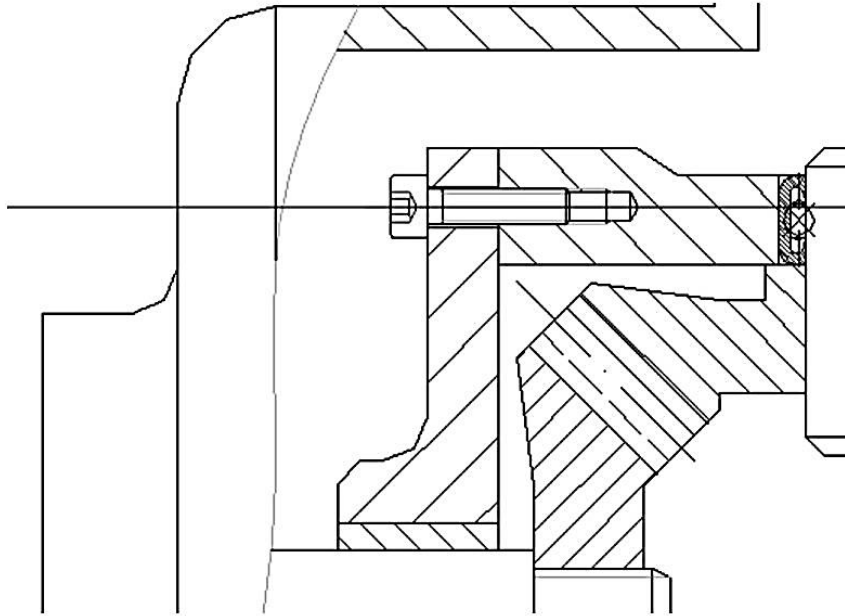
Drag Size:

*Drag the screw connection dynamically to size M4 x 16, and then click*

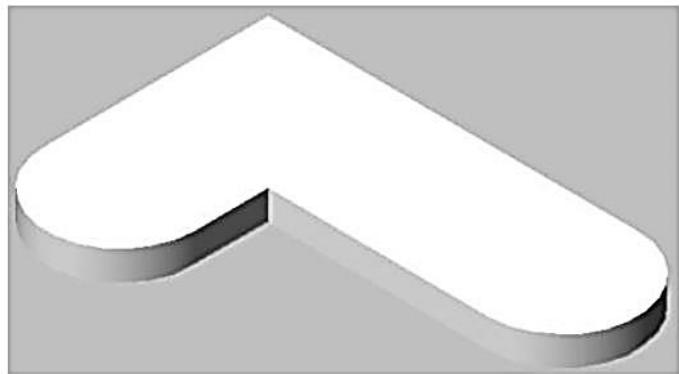
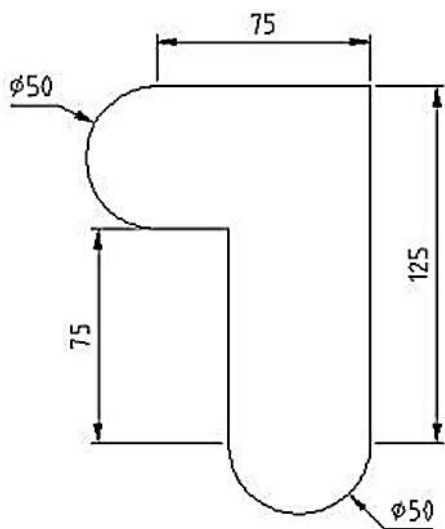
Drag Size: *Enter 12, press ENTER*

The screw connection is inserted with a specified a screw length of 16mm and a blind hole depth of 12 mm

**Note:-** During dragging, the size of the screw is shown as a tooltip and in the status bar, where the coordinates are usually displayed. The background is automatically hidden, and your drawing looks like this:

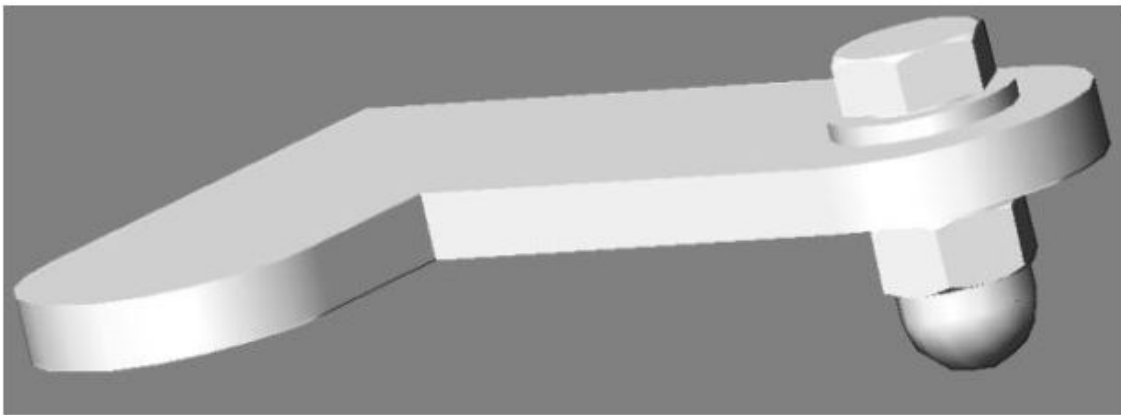
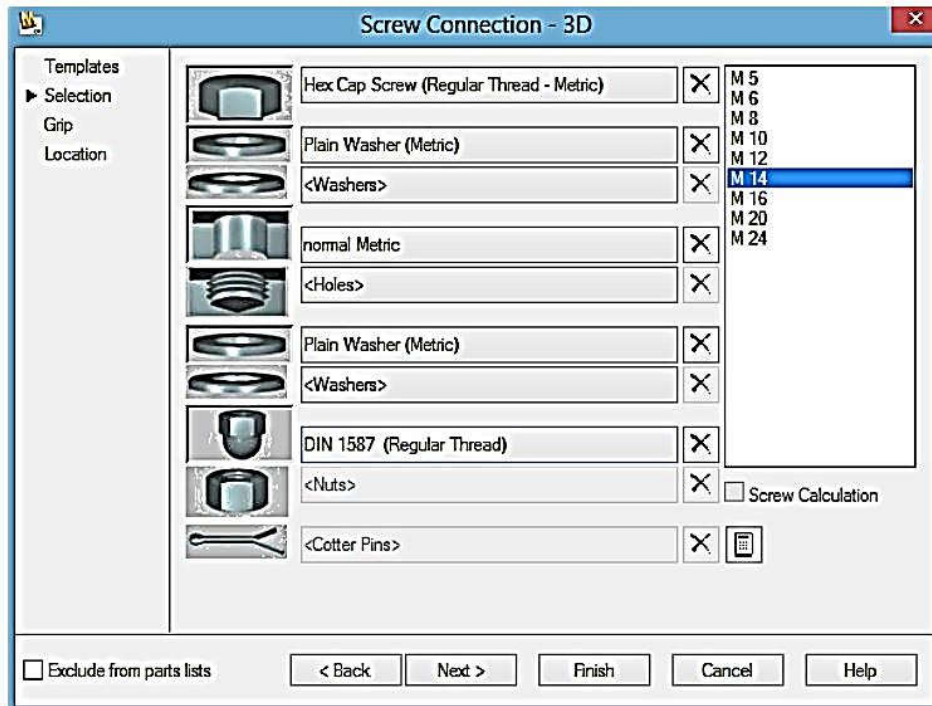


**Example 3:-** Do a 3D screw connection for this plate

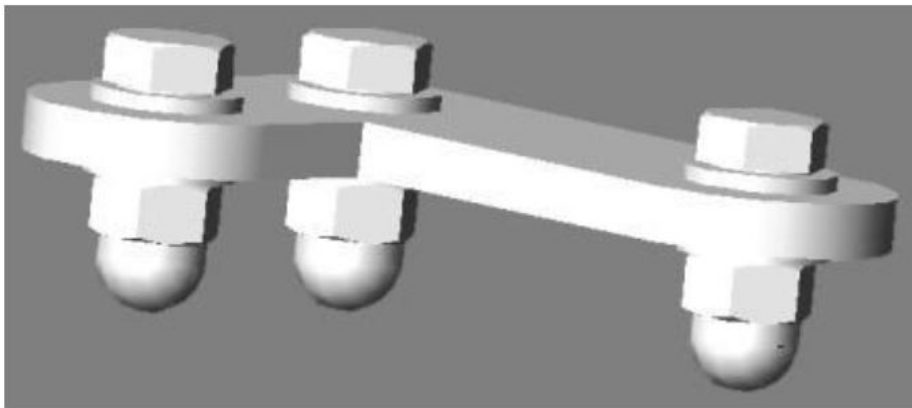


**Solution:-**

- 1- Draw the plate using given dimensions.
- 2- Make an extrude on the plate with a height of 10.
- 3- Use the date in the below picture.
- 4- In the insertion methods use Concentric method as mentioned above



**Exercise:-** Repeate Example 3 by using cyl Axial and Two edges.



## Shaft Generator

**Shaft Generator:-** Is a tool to draw rotationally symmetrical parts. A shaft is usually created from left to right using different sections. These sections are positioned automatically shaft generator one after the other. Additionally, any shaft section can be inserted, deleted, or edited.

### How to create a 2D or 3D Shaft by using MDT 2009 program?

1- Menu  $\Rightarrow$  content2D or 3D  $\Rightarrow$  Shaft Generator

3- In the command line write **AMSHAFT2D** or **AMSHAFT3D**

4-

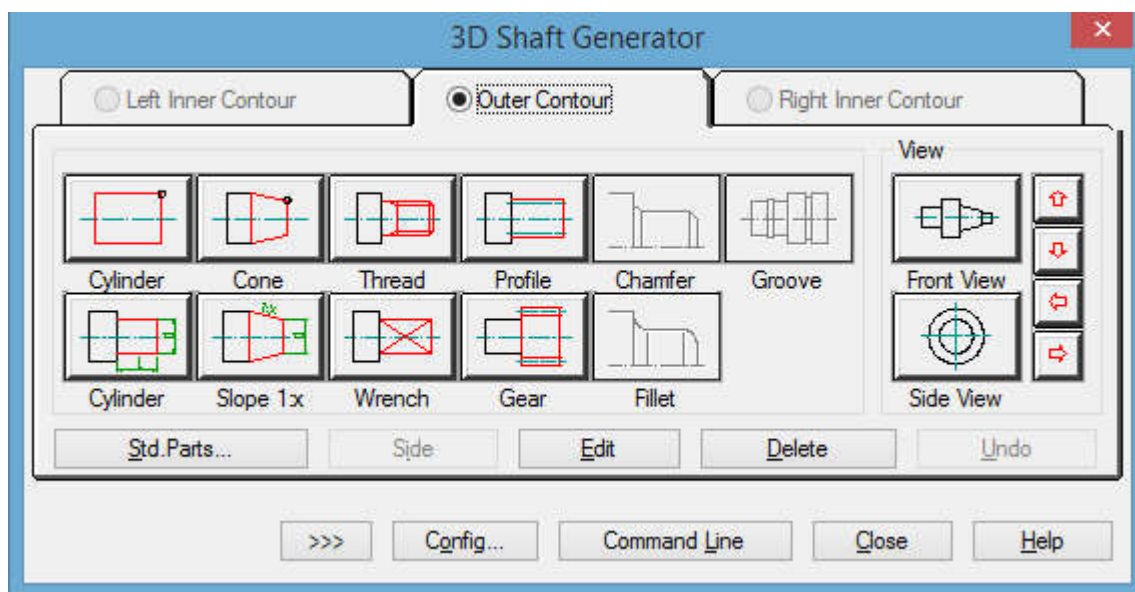
### COMMAND LINE OPTIONS:-

Specify start point or [Existing shaft]: *Specify a start point for a new shaft or select an existing shaft.*

Specify centerline endpoint: *Specify an endpoint for the centerline of the shaft.*

Enter shaft component name <Shaft1>: *Enter name of new shaft component. Predefined name is Shaft1.*

Specify point for new plane <parallel to UCS>: Select a point for a new plane, or press ENTER to set a new plan parallel to the current ucs. The 3D Shaft Generator dialog box is displayed



### **DIALOG BOX DEFINITIONS:-**

*3D Shaft Generator dialog box*

*Use the Shaft Generator dialog box to create and edit sections of a shaft.*

*Outer Contour tab*

#### **Cylinder (top)**

*Inserts a cylinder by dragging the contour.*

#### **Cylinder (bottom)**

*Inserts a cylinder using the keyboard to enter length and diameter dimensions.*

#### **Cone/Slope 1:x**

*There are two options in the Shaft Generator dialog box for creating conical shaft geometry: Slope and Cone. Both methods allow graphical and keyboard input, but the Slope method gives you greater accuracy control. You can specify the dimensions of conical geometry by responding to the command prompts or by using the dialog box.*

#### **Thread**

*Inserts threaded shaft segments into the shaft. When you select the Thread option, the Thread dialog box opens.*

#### **Profile**

*Inserts sectional profiles as shaft sections.*

#### **Wrench**

*Accesses the Wrench Opening dialog box.*

#### **Gear**

*Generates gears with involute teeth on a shaft segment.*

#### **Groove**

*Inserts grooves into a shaft section.*

#### **Chamfer**

*Chamfers shaft sections.*

#### **Fillet**

*Fillets shaft sections.*

#### **View**

*Front View*

*Changes a front view representation of a shaft to the side view.*

*Side View*

*Changes a side view representation of a shaft to the front view.*

**Arrows**

*Rotates the shaft incrementally by 15°.*

**Std. Parts**

*Displays the Select Group dialog box. Selected parts contain information that causes the shaft generator to draw the parts as a shaft segment.*

**Side**

*Changes the insertion direction from last segment to first and vice versa.*

**Edit**

*Modifies the particular sections of the shaft. This function may be considered as an internal power edit within the shaft generator.*

**Delete**

*Deletes a segment. The parts next to the deleted segment are moved by the length of the deleted section in the direction that is specified in the configuration.*

**Undo**

*Retraces the most recent step of the shaft generation. The inserted shaft sections can be removed, step by step, as long as the shaft generation has not been terminated.*

*>>>*

*Closes the dialog box temporarily so that you can look at the shaft.*

**Config**

*Adapts the routine, according to your requirements, to use the shaft generator.*

**Command Line**

*Displays all dialog box options on the command line.*

*Shaft Generator - Configuration dialog box*

*The Shaft Generator Configuration dialog box contains settings for the 3D Shaft Generator.*

*Initial State*

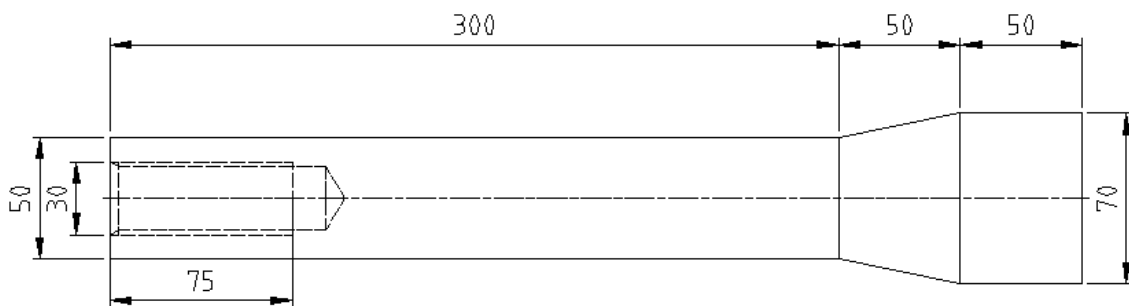
Operates from the Command line or via dialog box.

Representation of Gear

Displays one tooth of gear or the full gear geometry.

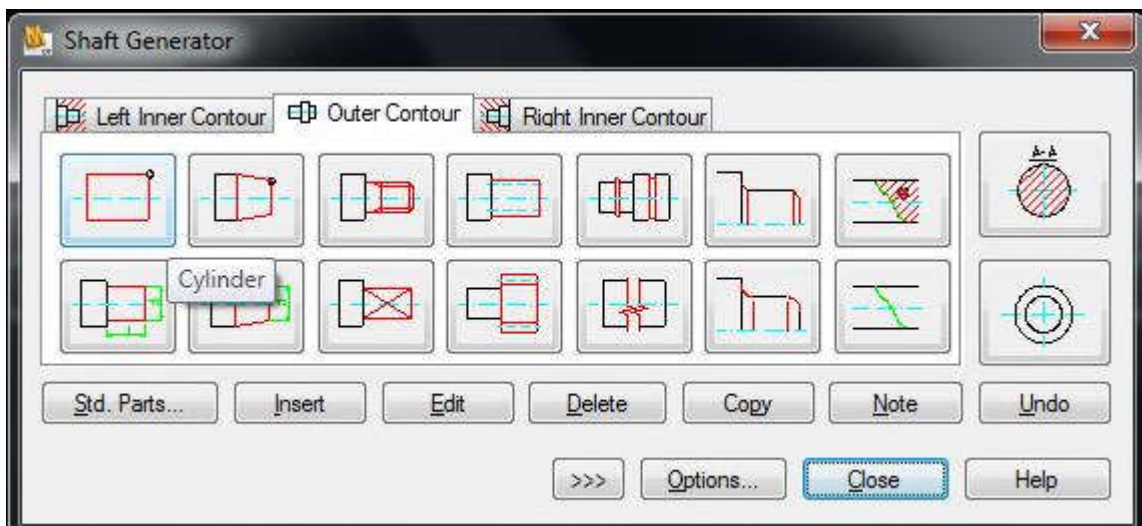
**Note:-** To set your changes to be default settings, select the Use As Default option.

**Example 2DSH:-** Create a 2D Shaft with a given dimensions as appear in figure below by using MDT 2009 program.



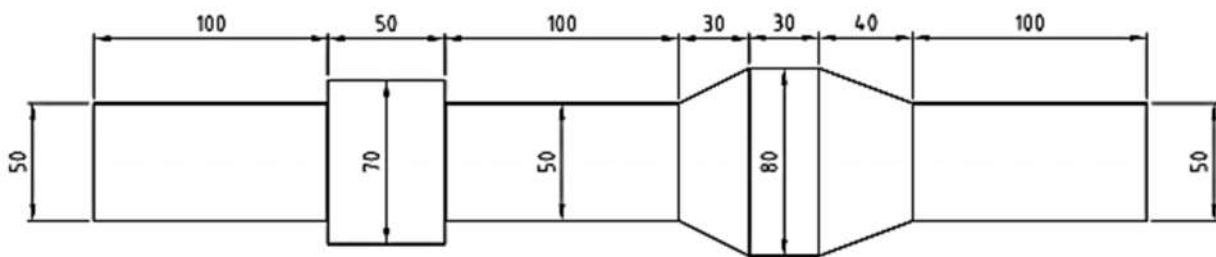
**Solution:-**

1. In the command line write **AMSHAFT2D** then press enter.
2. Specify starting point or select center line [New shaft]: *Click a point to specify an insertion point.*
3. Specify centerline endpoint: *Click a point on a horizontal axis.*
4. Then Shaft Generator dialog box will be appearing. To obtain the name of a button, place the cursor over a button. A tooltip showing the name of the button is displayed.
5. Inserts a cylinder using the point entry method.
6. Specify other corner point: 300, 25 (300 is Shaft length and 25 is Shaft radius).

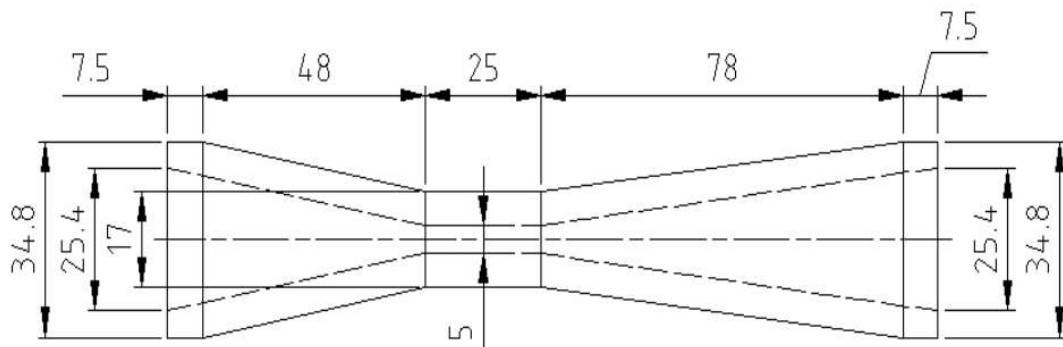


7. Then choose a Cone.
8. Specify other corner point: @50,35
9. After that click the bottom Cylinder.
10. Specify length <50>: 50
11. Specify diameter <70>: 70
12. Finally, in the dialog box change to the **Left Inner Contour** and click the **Thread**.
13. Choose Thread type of (M-ISO fine).
14. Specify length <50>: 75

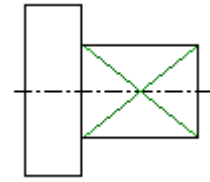
**Exercise:-** Creaa 2D shaft as shown in figure below by using MDT2009 program.



**Exercise:-** Create a 3D Venture meter in MDT2009 program as shown below.



## Wrench



*Insert a wrench opening:-*

*Wrench openings with sectional views for a two-sided, four-sided, or a six-sided segment of the shaft that fit a standard wrench are available from the Wrench opening dialog box.*

- 1. In the Shaft Generator dialog box, select the Wrench option.*
- 2. In the Wrench Opening dialog box, select the type of wrench to be inserted.*
- 3. In the corresponding dialog box, specify the geometrical values.*

Use the Wrench dialog box to specify a standard size for a wrench. Depending on the installed standard systems you can select several wrench types.

*Standard Size*

Represent standard sizes for the selected wrench type.

*Drag s*

Use the button to determine the size by dynamic dragging in the drawing.

*Length l=*

Enter a value for the segment length or select the Measure option on the context menu to determine the length by specifying two points in the drawing.

*Rotate*

Rotates the wrench from width across flats to width across corners.

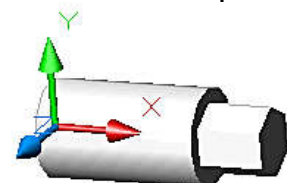
*Standard*

Opens Wrench opening dialog box.

**Note:-** The options diameter and length are only available for the two-sided wrench. If no value is entered in the Nominal diameter  $d$  field, an appropriate diameter from the database is attached to the width. After selecting a value in this field, this value is taken into consideration, in that it must be larger than the wrench opening.

**Insert a wrench opening to a 3D shaft:-**

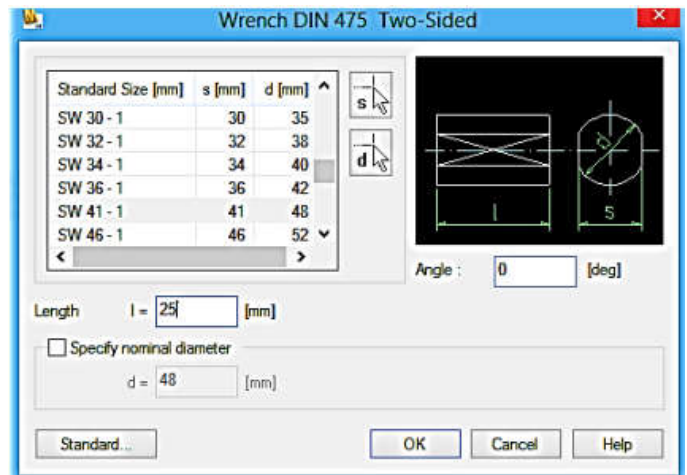
Wrench openings with sectional views for a two-sided segment, a four-sided segment, or a six-sided segment of the shaft that fit a standard wrench are available from the



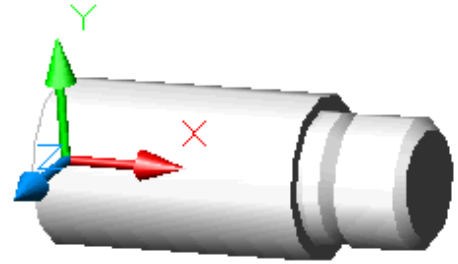
Wrench opening dialog box.

1. In the Shaft Generator dialog box, select the Wrench option.
2. In the Wrench Opening dialog box, select the type of wrench to insert.
3. In the corresponding dialog box, specify the geometrical values.

The program inserts the wrench after the last segment.



## Thread



Insert a thread to a 2D OR 3D shaft:-

1. In the Shaft Generator dialog box, select the Thread option.
2. In the Thread Selection dialog box, select the thread type to insert.
3. In the corresponding dialog box, you can specify the geometrical values for the section.

The program inserts the thread after the last segment.

Thread dialog box:-

Use the Thread dialog box to specify a standard size for a thread.

Depending on the installed standard systems you can select a thread type. The thread is provided with an undercut or a normal runout. For normal usage, the distance from the last full thread to the other shaft surface can be any length.

### Standard Size

**Represent standard sizes for the selected thread type.**

#### ***Drag d***

Use the button to determine the diameter by dynamic dragging in the drawing.

#### ***Length l=***

Enter a value for the length or select the Measure option on the context menu to determine the length by specifying two points in the drawing.

#### ***Undercut***

Selects thread with undercut.

#### ***dg***

Optional to enter a value for the general diameter.

*r*

Optional to enter a value for the radius of undercut.

*g1, g2*

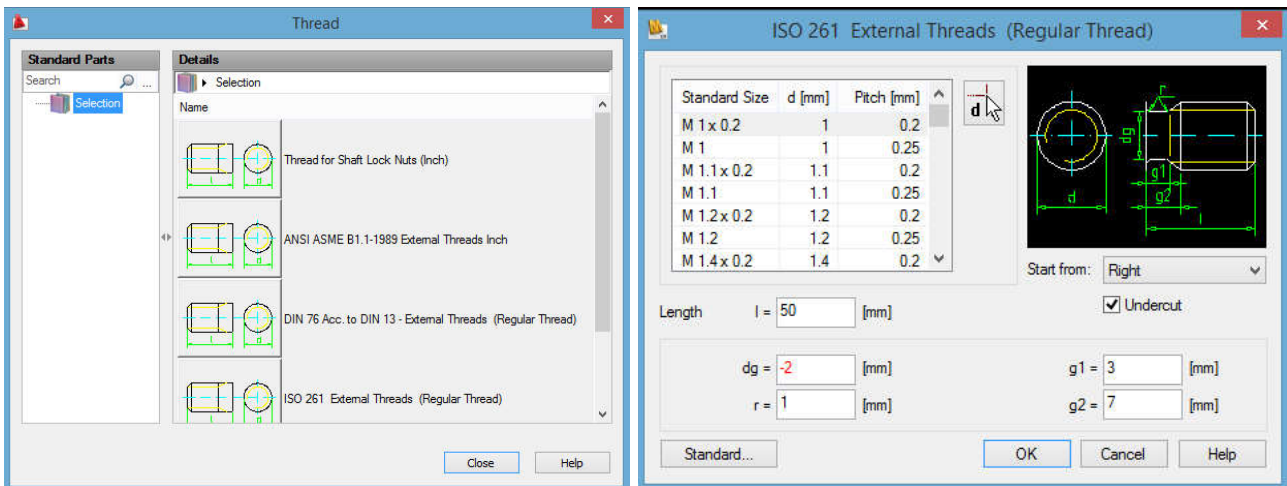
Optional to enter a value.

**Start from**

Start thread from right or left.

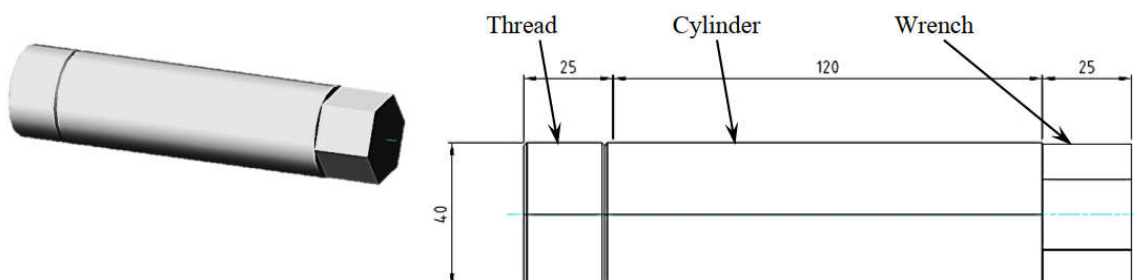
**Standard**

Opens the Thread dialog box.



Exercise:- Insert wrench and thread to a shaft as shown in figure below by using MDT2009 program.

Thread	Wrench
ISO 261 External Threads (Regular Thread)	DIN 475 six-sided (s = 34)



## Gear

**Gear:-** Any of several arrangements, especially of toothed wheels in a machine, which allow power to be passed from one part to another to control the power, speed, or direction of movement.

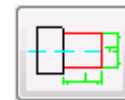
### Creating Cylindrical Shaft Sections and Gears:-

The shaft generator is configured. Now you want to generate the first shaft segments. Verify that the Outer Contour tab is selected. To create shaft segments

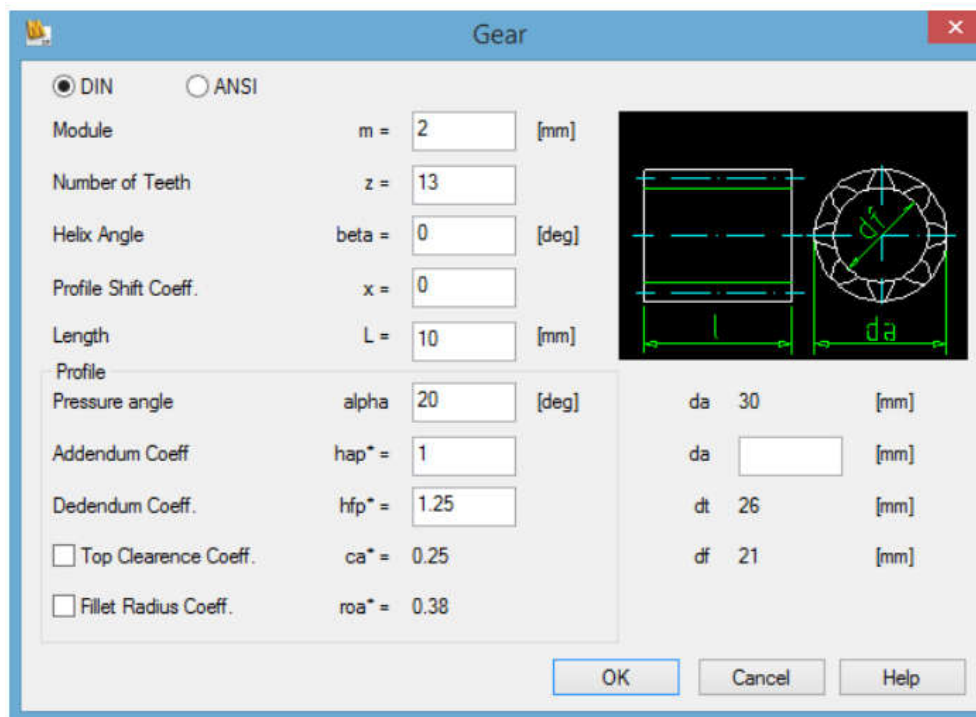
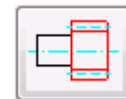
1) Click the lower cylinder button to define a cylinder section, and respond to the prompts as follows:-

Specify length <50>: *Enter 12, press ENTER*

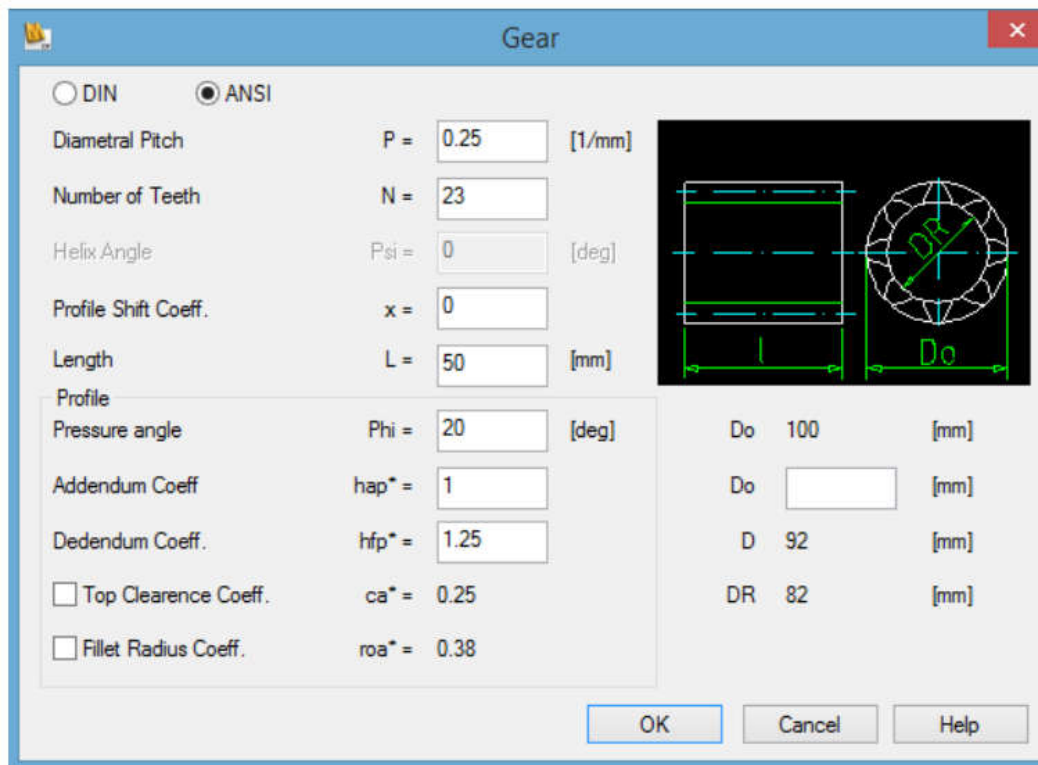
Specify diameter <40>: *Enter 20, press ENTER*



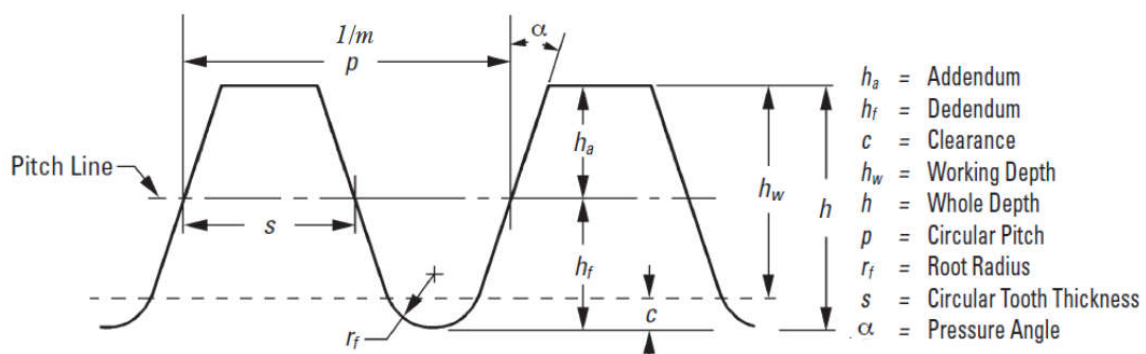
2) Click the gear button, and then enter the values for module, number of teeth, and length as shown in the following figure:-



**NOTE** Here, the DIN standard requires that you indicate the module. The ANSI standard requires the Diametral Pitch 1/module. You can switch between these two representations using the DIN and ANSI options.



Use the Gear dialog box to enter specific values for a gear for **DIN** or **ANSI**.



### **Module (DIN) – m**

Ratio of the pitch diameter to the number of teeth. Ordinarily, module is understood to mean ratio of pitch diameter in millimeters to the number of teeth. The English module is a ratio of the pitch diameter in inches to the number of teeth.

### **Diametrical Pitch (ANSI) – p**

The ratio of the number of teeth to the number of inches of pitch diameter - equals number of gear teeth to each inch of pitch diameter. Normal Diametral

Pitch is the diametral pitch as calculated in the normal plane and is equal to the diametral pitch divided by the cosine of the helix angle.

***Number of Teeth -  $z$  (DIN) /  $N$  (ANSI)***

Enter a value for the number of teeth.

***Pressure Angle -  $\alpha$  (DIN) /  $\Phi$  (ANSI)***

The angle between a tooth profile and a radial line at its pitch point. Standard Pressure Angles are established in connection with standard gear-tooth proportions. Most spur gears are cut to operate at pressure angles of 20 or 25°.

***Helix Angle -  $\beta$  (DIN) /  $\Psi$  (ANSI)***

The angle that a helical gear tooth makes with the gear axis at the pitch circle unless otherwise specified.

***Profile Shift or Addendum Modification -  $x$***

The Profile Shift will be taken to avoid undercuts by small number of teeth, to get a given center distance and to raise the load capacity. If the factor is positive the Addendum will be bigger and if the factor is negative the Addendum will be smaller, also the diameters will change.

Standard value:  $-0.7 < x < +0.7$

***Addendum Coefficient -  $h_{ap}^*$***

Distance between basic rack reference line and basic rack root line.

(Standard value  $h_{ap}^*=1$ )

***Dedendum Coefficient -  $h_{fp}^*$***

Distance between basic rack reference line and basic rack head line.

(Standard value  $h_{fp}^*=1.25$ ).

***Top Clearance Coefficient -  $ca^*$***  Coefficient of distance between outside diameter of first gear and root diameter of second gear in pair.

(Standard value  $ca^* = 0.25$ )

***Fillet Radius Coefficient -  $roa^*$***

coefficient of fillet radius at rack cutter. (Standard value  $ca = 0.25$ )

### **Length**

Determines the length of the gear.

**da (DIN)/ do (ANSI)**

Outside (tip) diameter

**dt (DIN)/ D (ANSI)**

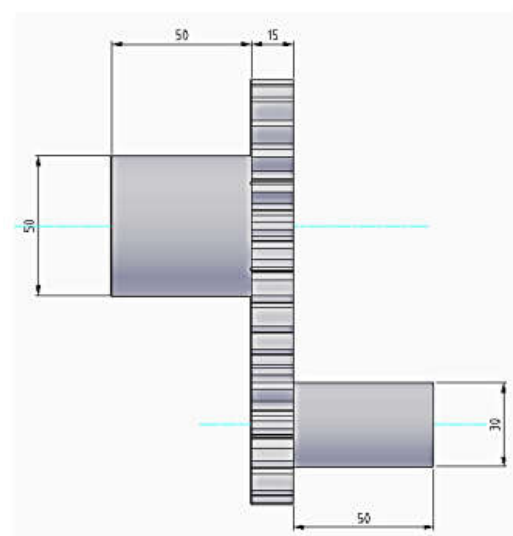
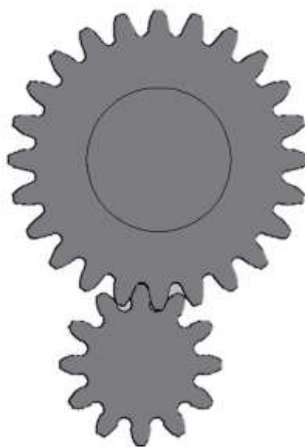
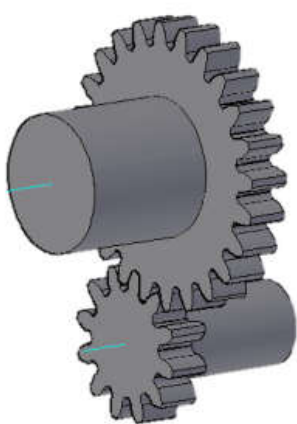
The diameter of the pitch circle. In parallel shaft gears the pitch diameters can be determined directly from the center distance and the numbers of teeth by proportionality. The pitch circle is the radius which is equal to the distance from the gear axis to the pitch point.

**df (DIN)/ DR (ANSI)**

Root diameter: Diameter of the root circle.

**Example Gear:-** Draw the assembly drawing for gears as shown in the figure by using these specifications:-

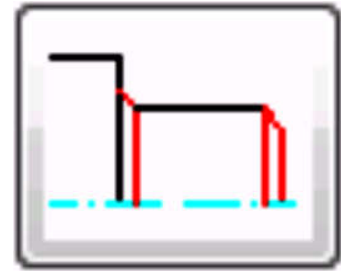
<b>Gear A</b>		<b>Gear B</b>	
<b>Module</b>	4	<b>Module</b>	4
<b>No. of Teeth</b>	24	<b>No. of Teeth</b>	12
<b>Length</b>	15	<b>Length</b>	15
<b>da</b>	104	<b>da</b>	56



## Chamfer and Fillet

**Chamfer:-** A beveled surface between two faces or surfaces. (For example, Cylindrical shaft edges).

**Fillet:-** A curved transition from one part face or surface to another. The transition cuts off the outside edge or fills in the inside edge (Such as, Cylindrical shaft edges).



In order to apply a chamfer and a fillet to the shaft **by using MDT program (2D OR 3D)**, **Follow** these steps:-

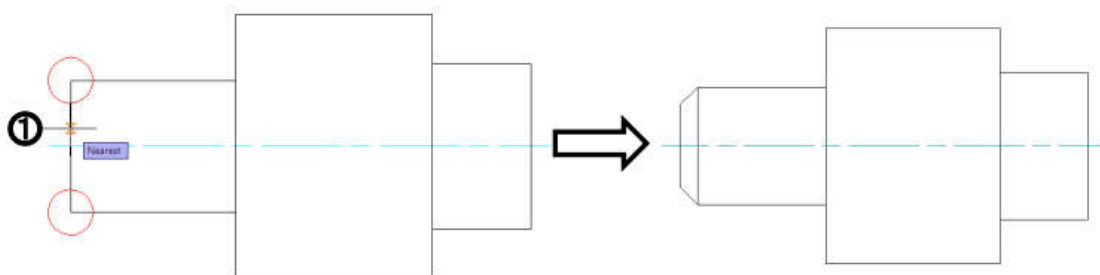
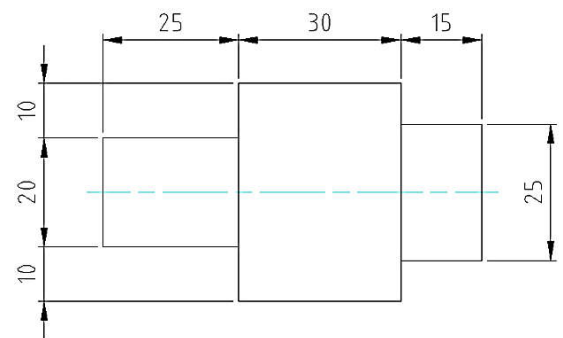
1) **Draw** a cylindrical shaft as shown in figure:-

2) **Click** the Chamfer button in shaft generator dialog box to apply a chamfer to a shaft section, and then respond to the prompts as follows:-

Select object: Select the leftmost cylinder section ①

Specify length (max. 25) <2.5>: Enter 3, press ENTER

Specify angle (0-73) or [Distance] <45>: Enter 45, press ENTER



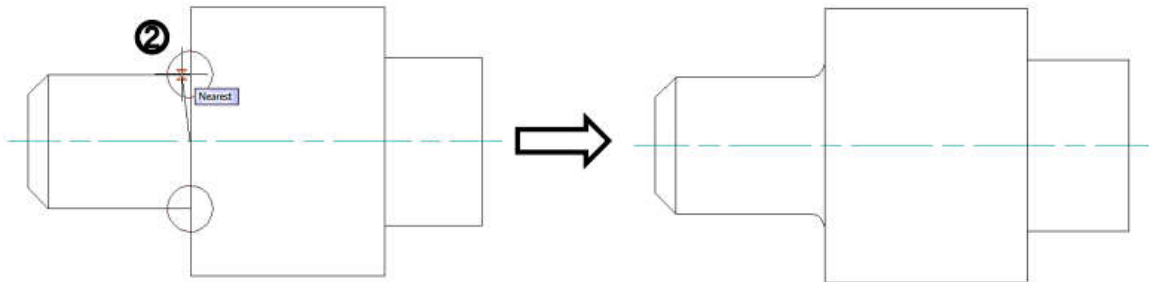
**Note:-** The max. length of Chamfer and Chamfer angle are change according to the shaft diameter change.

3) In order to create a **Fillet** Click the Fillet button and then respond to the prompts as follows:-

Select object: *Select the first cylindrical section near to the second cylindrical section*

②

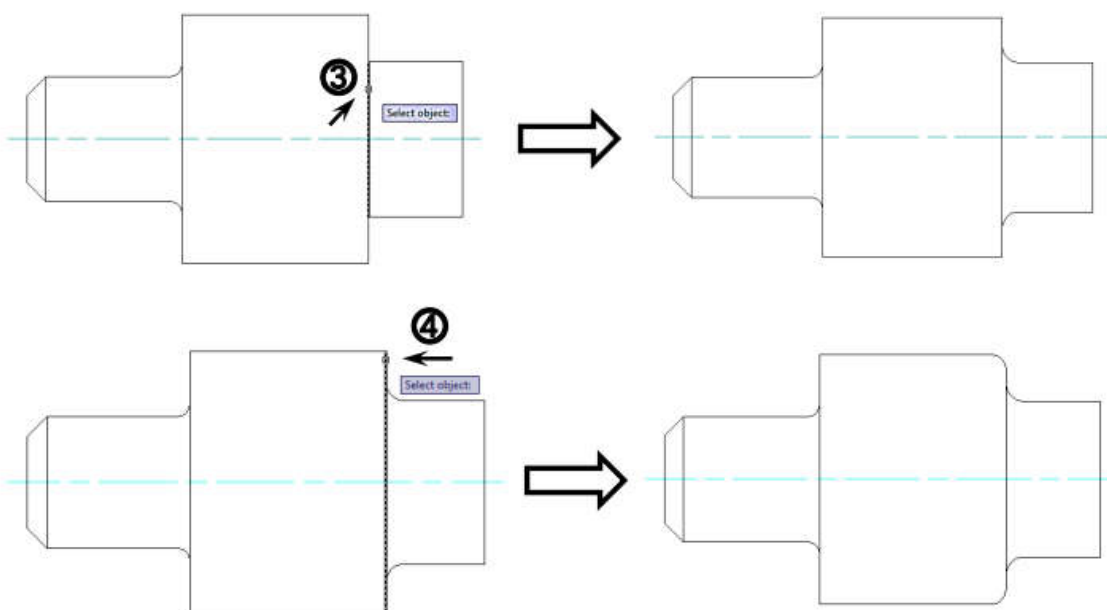
Enter radius (max. 10.00) <2.5>: *Enter 2, press ENTER*



**Note:-**

- The max. radius of Fillet is changes according to the shaft diameter change.
- The fillet is applied to the edge of the selected section that is closer to the selected point.

For example, if you click the line ( Point ③ ) as shown below to create a fillet, the fillet will be created to the inside edge, else if you click the line ( Point ④ ) the fillet will be created to the outside edge.





## Shaft Component

To add shaft components to a 2D OR 3D shaft:-

1. In the Shaft Generator dialog box, select the Std. Part... option.
2. In the Standard Parts Selection dialog box, select the part to insert.
3. Depending on the standard part you selected, further dialog boxes open.

**Note:-** Based on part information, the Shaft Generator draws the part (also in the sectional view)

## Roller Bearing

A **Bearing** is a machine element that constrains relative motion and reduces friction between moving parts to only the desired motion. The design of the bearing may, for example, provide for free linear movement of the moving part or for free rotation around a fixed axis; or, it may prevent a motion by controlling the vectors of normal forces that bear on the moving parts. Many bearings also facilitate the desired motion as much as possible, such as by minimizing friction. Bearings are classified broadly according to the type of operation, the motions allowed, or to the directions of the loads (forces) applied to the parts. Two main types of Roller Bearing can be drawn in MDT program:-

1) **Radial roller bearing:-** (a) Closed (Needle). (b) Ball Bearing. (c) Airframe Roller Bearing. (d) Tapered Roller Bearing.

2) **Axial roller bearing:-**

- (a) Thrust Ball Bearing.
- (b) Thrust Roller Bearing.
- (c) Thrust Spherical Roller Bearing.
- (d) Thrust Needle Roller Bearing.
- (e) Thrust Tapered Roller Bearing.



**In order to insert a Roller Bearing on a shaft by using MDT 2009 program, use these steps:-**

**1-** Use to define the shaft AMSHAFT3D

Respond to the prompts as follows:-

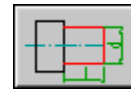
Specify start point or [Existing shaft]: *Select a point in the drawing area*

Specify centerline endpoint: *Drag a line to the right, and select a point*

Specify point for new plane <parallel to UCS>: *Press ENTER*

The 3D Shaft Generator dialog box is displayed. Define the shaft contour, starting with the segment at the left, and working to the right.

**2-** Verify the Outer Contour tab is selected, and then choose the lower Cylinder icon.

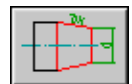


Respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <50>: Enter 10

Specify diameter or [Associate to/Equation assistant] <40>: Enter 74

**3-** Choose the Slope icon, and respond to the prompts as follows:



Specify length or [Dialog/Associate to/Equation assistant] <10>: Enter 7

Specify diameter at start point or [Associate to/Equation assistant] <74>:

Press ENTER

Specify diameter at end point or [Slope/aNgle/Associate to/Equation assistant] <72>:

Enter 48

**4-** Choose the lower Cylinder icon again, and respond to the prompts as follows:

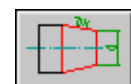
Specify length or [Associate to/Equation assistant] <7>: Enter 20

Specify diameter or [Associate to/Equation assistant] <48>: Enter 40



**5-** Choose the Slope icon again, and respond to the prompts as follows:

Specify length or [Dialog box/Associate to/Equation assistant] <20>: Enter 10



Specify diameter at start point or [Associate to/Equation assistant] <40>: Press ENTER

Specify diameter at end point or [Slope/aNgle/Associate to/Equation assistant] <36>: Enter 32

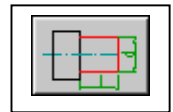
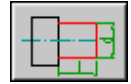
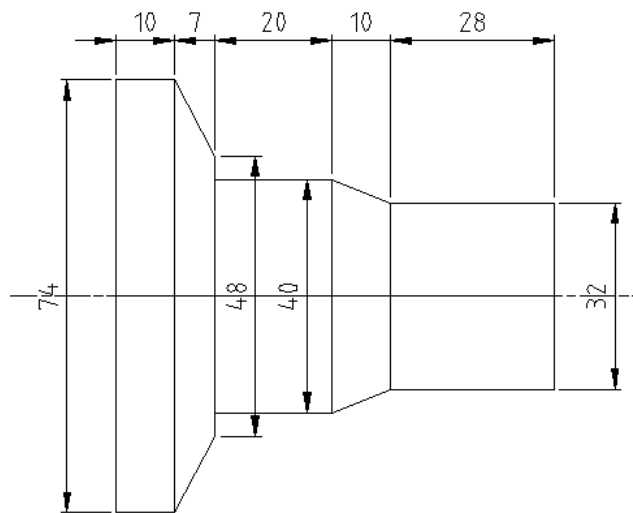
6- Choose the lower Cylinder icon once more, and respond to the prompts as follows:

Specify length or [Associate to/Equation assistant] <10>: Enter 28

Specify diameter or [Associate to/Equation assistant] <32>: Press ENTER

You have created a shaft consisting of three cylindrical and two conical segments.

Your drawing should look like this.



⇒ **To add a bearing to a shaft**

7- *MENU* ⇒ Content 3D ⇒ Shafts Components ⇒ Roller Bearings

OR in the command line write **AMROLBEAR3D**

*Options* Use the Select a Roller Bearing dialog box to select the type of bearing and the view to insert. Depending on the selected view, there are differences in the insertion process:-

In the Select a Part dialog box, choose Roller Bearings ► Radial ► ISO 355.

Respond to the prompts as follows:-

Select cylindrical face: *Select a point on the third shaft segment* ①

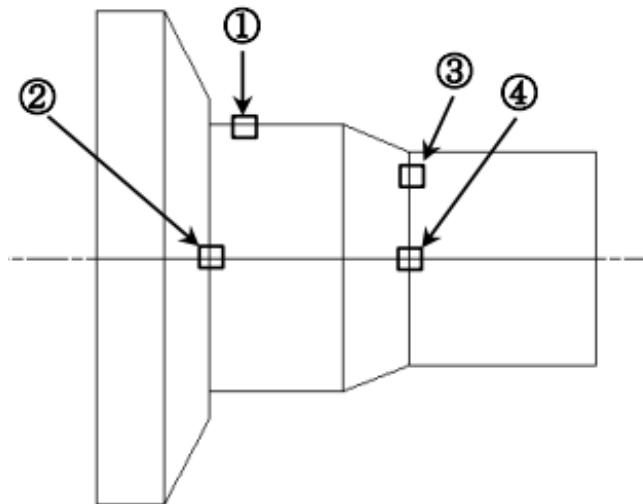
Specify location on cylindrical face [Line/Plane]: *Select the left plane of the third shaft segment* ②

Enter distance from base plane [Associate to/Equation assistant] <0>: *Press ENTER*

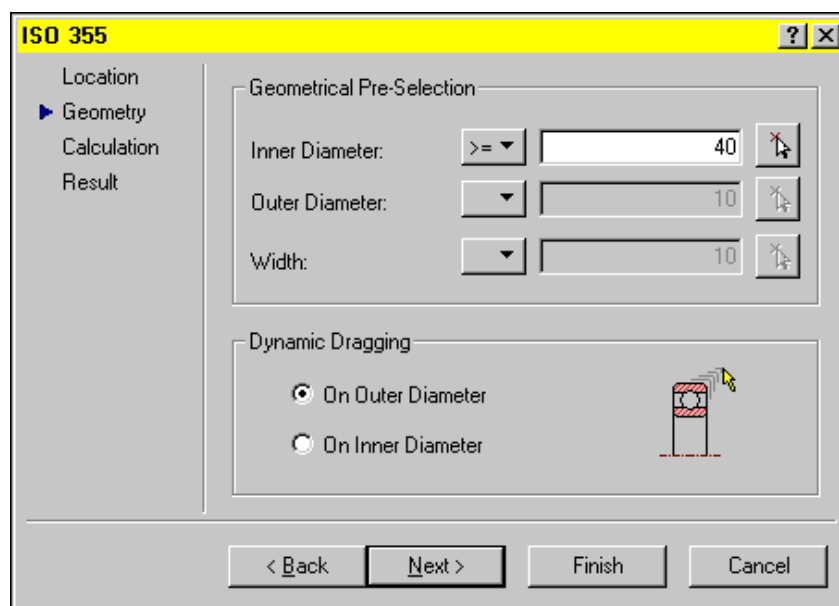
Choose insertion direction [Flip/Accept] <Accept>: *Press ENTER*

In the ISO 355 dialog box, verify Geometry is selected and specify: Inner Diameter: 40

Choose Next to continue. You use the ISO 355 dialog box for the bearing calculation.



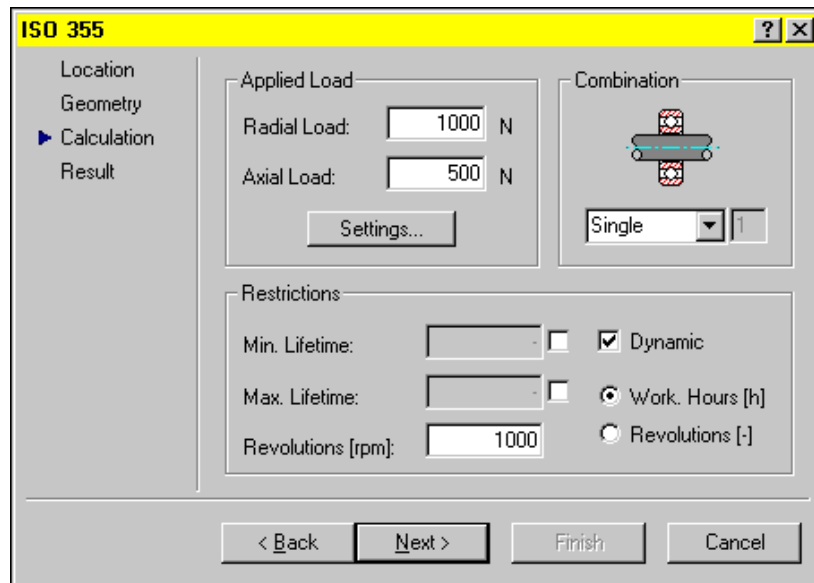
In the ISO 355 dialog box, verify Geometry is selected and specify: Inner Diameter: 40



Choose Next to continue.

You use the ISO 355 dialog box for the bearing calculation.

In the ISO 355 dialog box, verify Calculation is selected and specify the values as shown below.

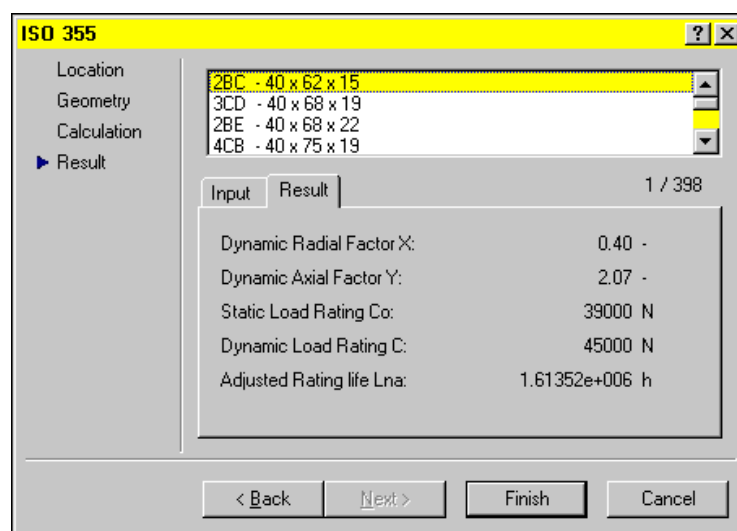


By choosing dynamic calculation, Mechanical Desktop is calculating the adjusted rating life of the bearing.

Choose Next.

The possible bearings are calculated and the results are displayed in the ISO 355 dialog box in Result. Two tabs are provided so you can check both your input and results.

In the ISO 355 dialog box, select the Result tab, and then select 2BC - 40 x 62 x 15.



Choose Finish.

⇒ **To add the second bearing**

In the 3D Shaft Generator dialog box, choose Std. Parts.

In the Select a Part dialog box, choose Roller Bearings ► Radial ► ISO 355.

Respond to the prompts as follows:-

Select cylindrical face: *Select a point on the fifth shaft segment* ③

Specify location on cylindrical face [Line/Plane]: *Select the right plane of the fifth shaft segment* ④

Enter distance from base plane [Associate to/Equation assistant] <0>: *ENTER 25*

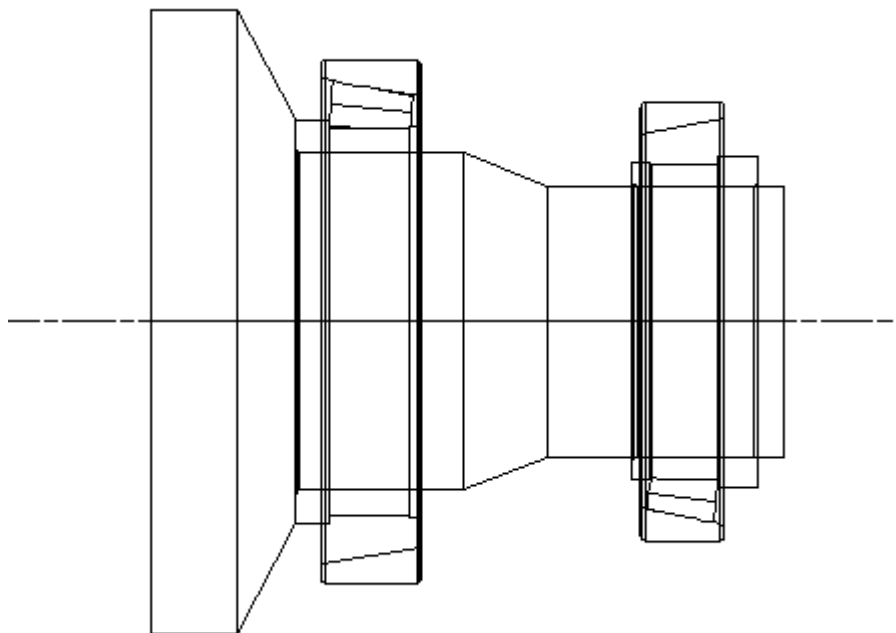
Choose insertion direction [Flip/Accept]: *Enter f*

Use the default values for the calculation, and select the bearing ISO 355

2BD - 32 x 52 x 14.

Choose Finish.

Your drawing should look like this.



## Bearing calculation

**Bearing calculation:-** Calculates limiting value, dynamic and static load rating, dynamic and static equivalent load, and fatigue life in revolutions and hours.

**Dynamic calculation:-** Calculation required for a revolving bearing. The result is the Adjusted Rating Life. This is the life associated with 90% reliability with contemporary, commonly dynamic calculation used material, and under conventional operating conditions. With the number of revolutions you get the life in working hours.

⇒ Use AMBEARCALC to perform calculation on bearings. You can determine the following bearing characteristics:

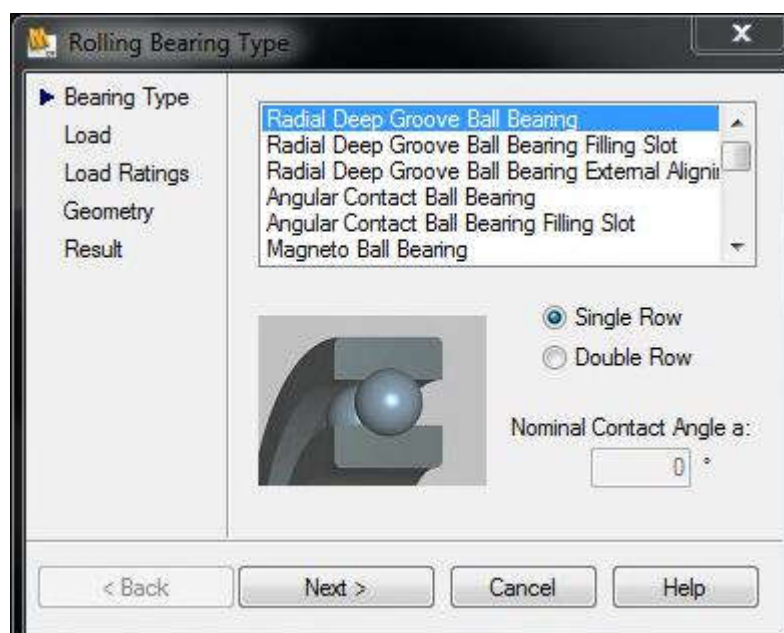
- Limiting value.
- Dynamic and static load rating.
- Dynamic and static equivalent load.
- Fatigue life, in revolutions and hours.

⇒ **To perform a bearing calculation**

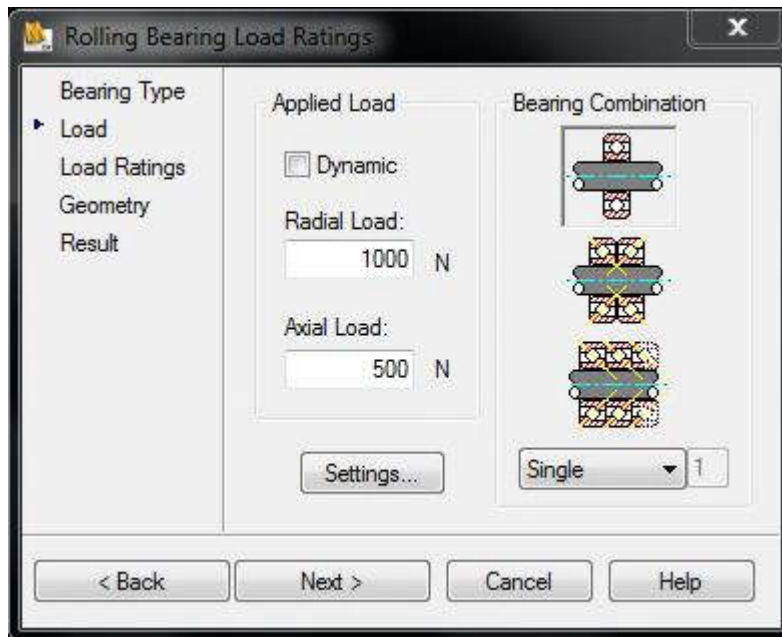
You can perform a bearing calculation on existing bearings in your drawing, or as a stand-alone calculation. The general procedure is the same in both cases.

1. In the command line, enter AMBEARCALC.

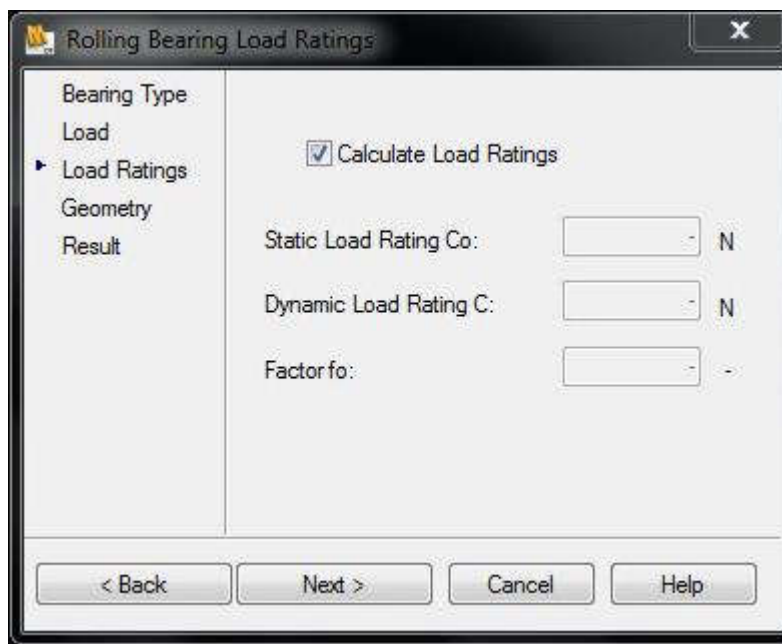
2. In the Rolling Bearing Type dialog box, select the type of bearing. Choose Next.



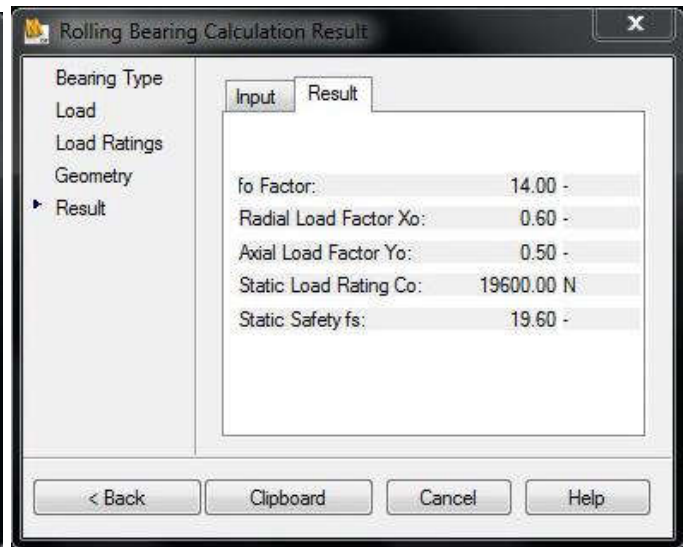
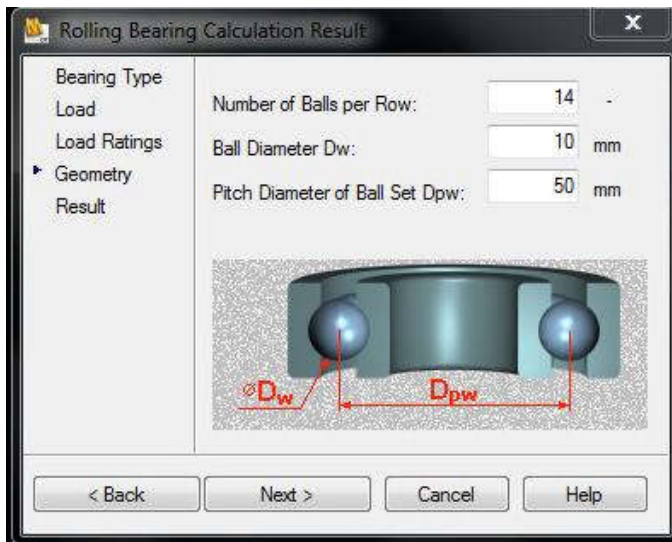
3. In the Rolling Bearing Load Ratings dialog box, specify the type of loads and the bearing combination. Choose Next.



4. In the Rolling Bearing Load Ratings dialog box, select Calculate Load Ratings. Choose Next.



5. In the Rolling Bearing Calculation Result dialog box, make the necessary entries. Choose Next to display the result of the calculation.



# Keys

In mechanical engineering, a **key** is a machine element used to connect a rotating machine element to a shaft. The key prevents relative rotation between the two parts and enables torque transmission. A key is used for temporary fastening. For a key to function, the shaft and rotating machine element must have a keyway, also known as a keyseat, which is a slot or pocket the key fits in. The whole system is called a keyed joint. A keyed joint still allows relative axial movement between the parts.

Use **AMSHAFTKEY2D** to insert parallel keys, woodruff keys or hubs from the installed standard parts.

## Options:-

You use the Please Select a Key or Hub dialog box to select the type and view to insert.

When you select a woodruff key, you can determine where you want to insert the key (on mid of cylindrical shaft or mid of conical shaft).

When you select a parallel key, you can determine on which type of shaft you want to insert the key (on mid of shaft or on end of the shaft).

## COMMAND LINE OPTIONS:-

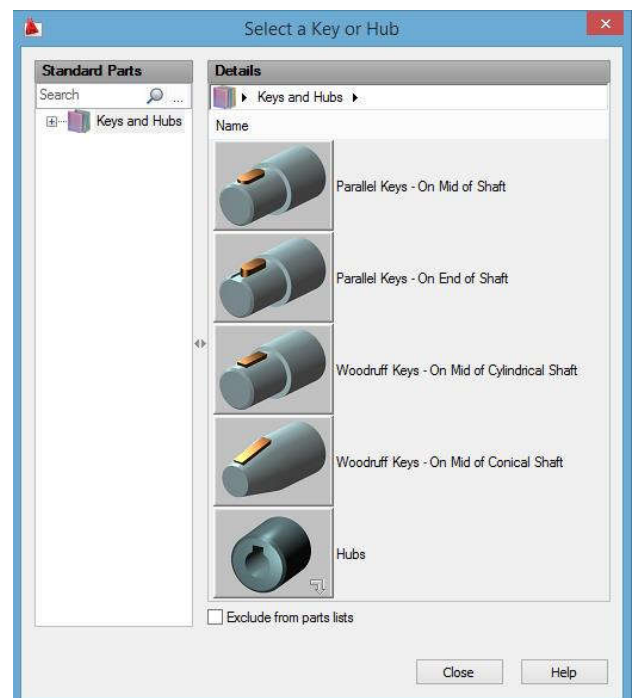
### *Front View/Top View*

Specify insertion point on shaft contour:

*Select a point on the shaft contour to specify the starting point of the key.*

Select centerline of shaft: Select the centerline of the shaft. In the Visibility dialog box, you can decide whether you want the part to be drawn.

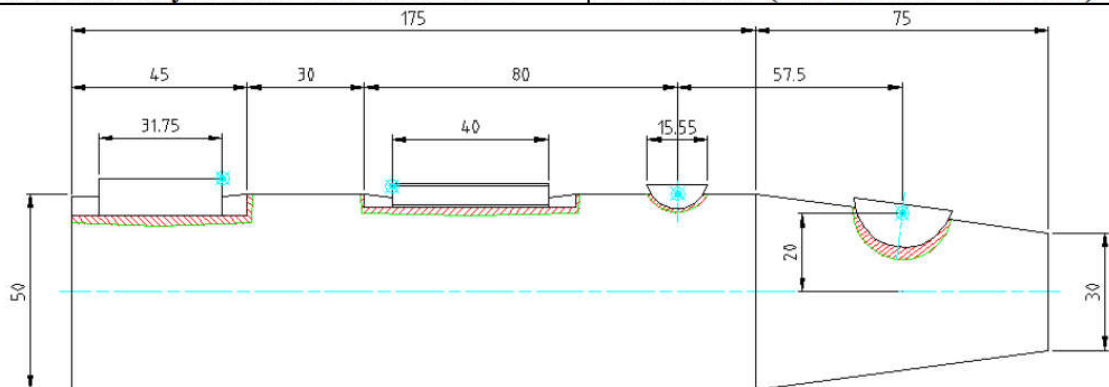
Drag size: *Drag the size of the key.*



**Example 1KEY:**

Draw the various types of key attached to a 2D shaft shown in Figure.

Key type	Details
Parallel key on end of shaft	Rectangular
Parallel key on mid of shaft	ISO 2491-B
woodruff key on mid of cylindrical shaft	Full-radius (ANSI B17.2 - No. 605)
woodruff key on mid of conical shaft	Full-radius (ANSI B17.2 - No. 608)

**Solution:-**

Firstly, draw the cylindrical and cone shaft. Then choose Std. Parts from shaft generator box and click on *Parallel key on end of shaft* choose.

From the dialog box of *Parallel key on end of shaft* choose front view and then choose type of key (Rectangular) to draw it according to these steps:-

Specify insertion point on shaft contour:- *Enter a point where you want to insert the key (45,25).*

Direction to [Left/Right]:- *L*

Drag size:- *31.75*

After that the key would be drawn. In order to draw other keys use same steps.

**To insert a 3D parallel/ woodruff key:-**

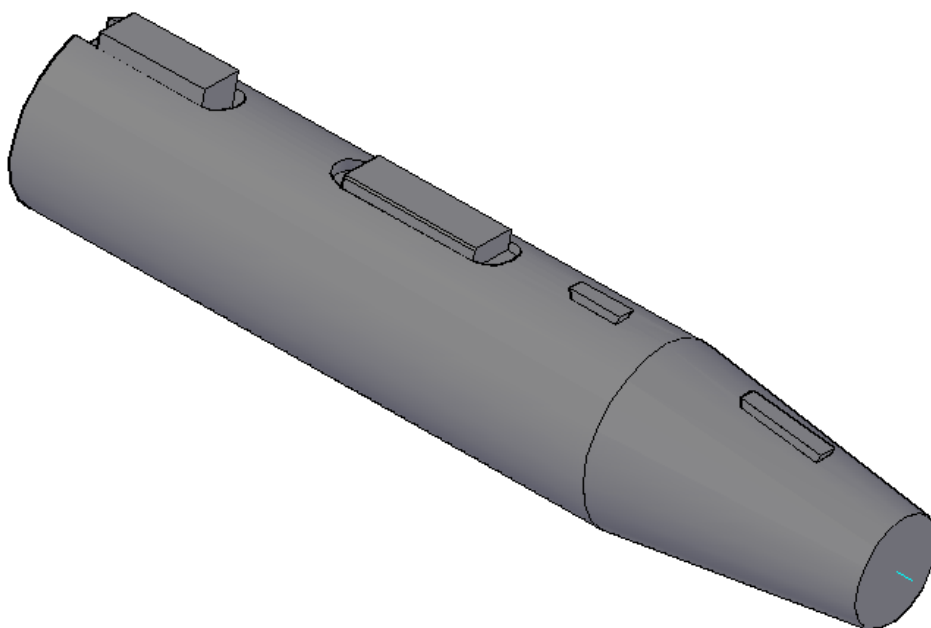
There are two ways to insert a woodruff/parallel key. Launch the

**AMSHAFTKEY3D** command or insert a parallel/woodruff key with the 3D Shaft Generator.

1. At the Command prompt, enter AMSHAFTKEY3D. In the 3D Shaft Generator dialog box, select the Std.Parts... option.
2. In the dialog boxes, select the key type.
3. Select the cylindrical face where you want to insert the key (1).
4. Specify the position for the key in an existing shaft section by dragging (2) or choose  
Line/Plain.
5. Enter a value for distance from base plane or  
press ENTER to accept.
6. Specify one of the following insertion methods:-
  - ❖ Angle to plane or edge.
  - ❖ Parallel to line.
  - ❖ Normal to plane.
  - ❖ Parallel to plane.

**Exercise:-**

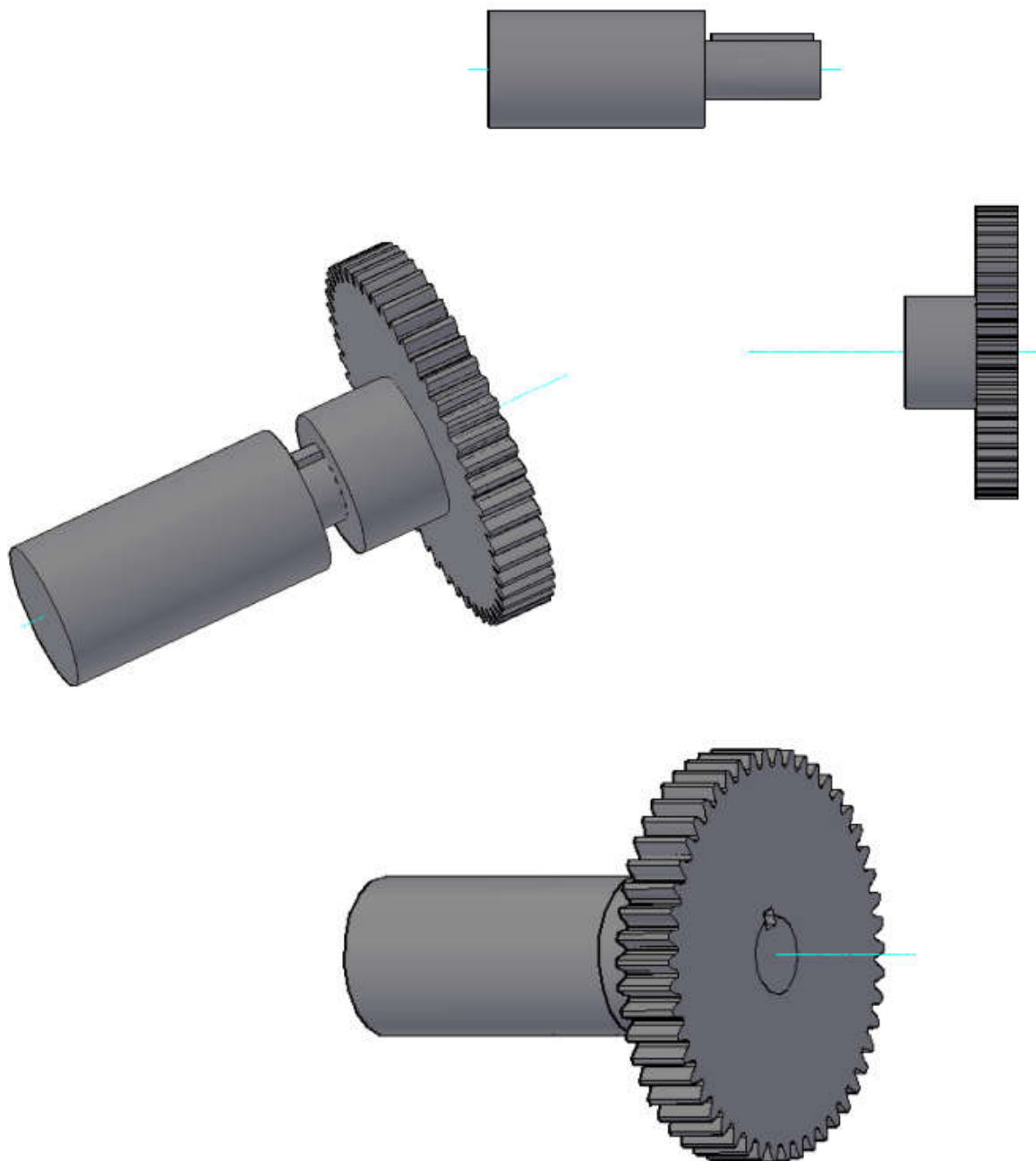
Draw the various types of key attached to a 3D shaft shown in Figure. (Use same standard given in Example 1KEY).



**Example 2KEY:-**

Draw the two parts using the dimensions listed in table and then connect them

Part 1	Part 2
Cylinder (L = 75 & Di = 40)	Cylinder (L = 25 & Di = 40)
Cylinder (L = 40 & Di = 20)	Gear (m = 2, z = 50 & L = 15)
Key (Square)	Hub (Square)



## Seals

### What is the purpose of a Rotary Shaft Seal?

Rotary shaft seals are used throughout many industries in a variety of different application conditions. These conditions can vary from high-speed shaft rotation with light oil mist, to a low speed reciprocating shaft in muddy environments.

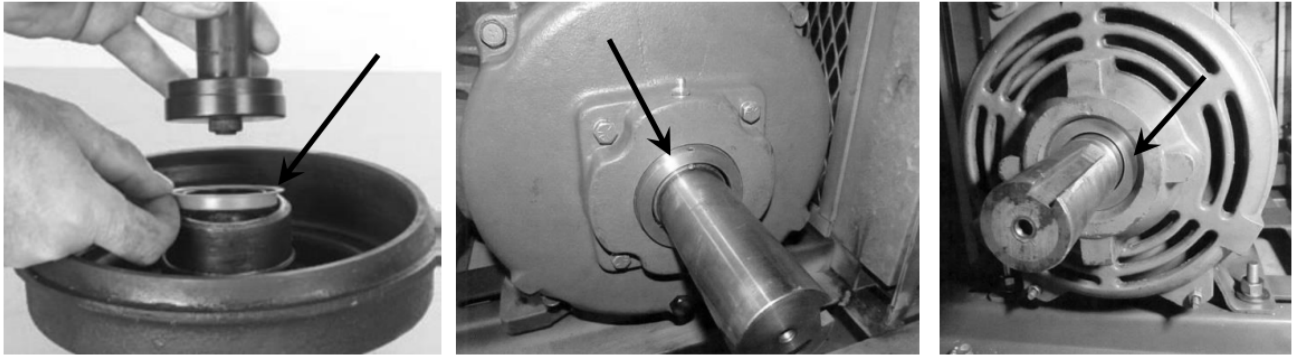
Rotary shaft seals can be required to seal lube oil in high-speed crankshaft applications or for gasoline and diesel engines that operate in conditions ranging from the tropics to the arctic; in submarines, oil tankers, windmills, steel mills, paper mills, refineries, automobiles and plant machinery.

In rotating applications, the seal can also impact the service life of indirect components such as mechanical seals, couplings, pulleys or other in-line coupled equipment. If a seal allows the system lubricant to run below safe levels, or allows foreign materials to enter the bearing cavity, the bearing will soon begin to show signs of failure. As the bearing fails, vibration from excessive shaft runout will be transferred to all other in-line components and will shorten their service life as well. Additionally, seals can also be used to separate two different fluids, to retain internal pressure or exclude an external pressure.

### How do Rotary Shaft Seals work?

Rotary shaft seals work by squeezing and maintaining the lubricant in a thin layer between the lip and the shaft. Sealing is aided by the hydrodynamic action caused by the rotating shaft which creates a slight pump action. The amount of frictional heat that is generated in an application is dependent on a combination of many operating parameters: shaft surface, internal pressure, operating speed, lubricant type, lubricant levels, lip geometries and lip materials are just a few conditions that should be considered, and it is important to note that these conditions are interactive.





Use **AMSEALS2D OR AMSEALS3D** to insert seals on shaft from the installed standard parts. For Example, to insert 3D Shaft Seal use these steps:-

- 1- Content 3D  $\Rightarrow$  Shafts/Components  $\Rightarrow$  Seals **OR** in the command line writes AMSEALS3D.
- 2- Use the Select a Shaft Seal dialog box to select the type to insert. Depending on the selected seal.
- 3- Select cylindrical face: *Click the edge of the area where you want to insert the standard part.*
- 4- Select position on cylinder or cone [Line/Plane]: *Specify the position by moving the cursor.*
- 5- Enter distance from base plane [Associate to/Equation assistant]: *Enter a value or press ENTER.*
- 6- Drag size [Dialog]: *Drag the size of the standard part by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table.*

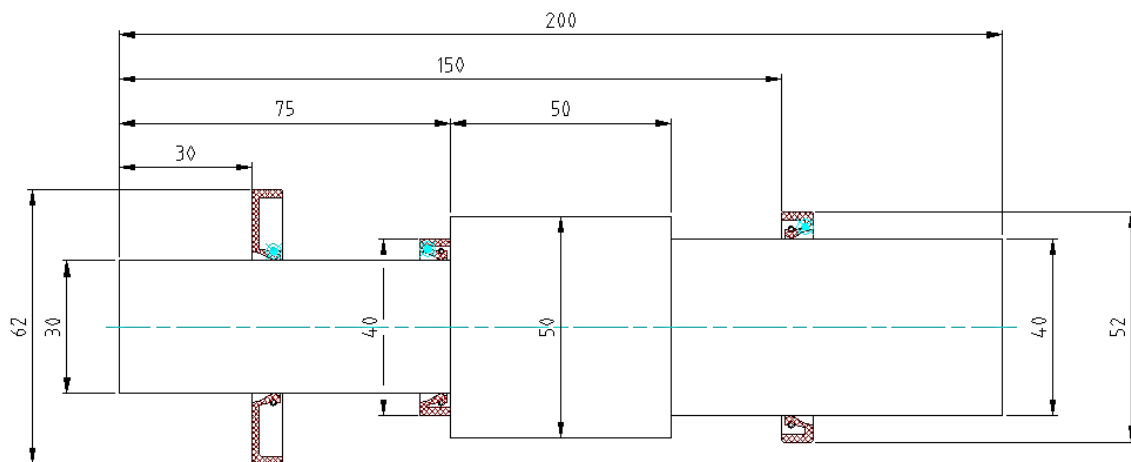
**Example 2DSeal:-** Insert the (DIN 3760 - A) Seals on the shaft as shown in Figure Use **AMSEALS2D OR AMSEALS3D** to insert seals on shaft from the installed standard parts. For Example, to insert 3D Shaft Seal use these steps:-

- 1- Content 3D  $\Rightarrow$  Shafts/Components  $\Rightarrow$  Seals **OR** in the command line writes AMSEALS3D.
- 2- Use the Select a Shaft Seal dialog box to select the type to insert. Depending on the selected seal.
- 3- Select cylindrical face: *Click the edge of the area where you want to insert the standard part.*
- 4- Select position on cylinder or cone [Line/Plane]: *Specify the position by moving the cursor.*

5- Enter distance from base plane [Associate to/Equation assistant]: *Enter a value or press ENTER.*

6- Drag size [Dialog]: *Drag the size of the standard part by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table.*

**Example : 2D Seal:-** Insert the (DIN 3760 - A) Seals on the shaft as shown in Figure



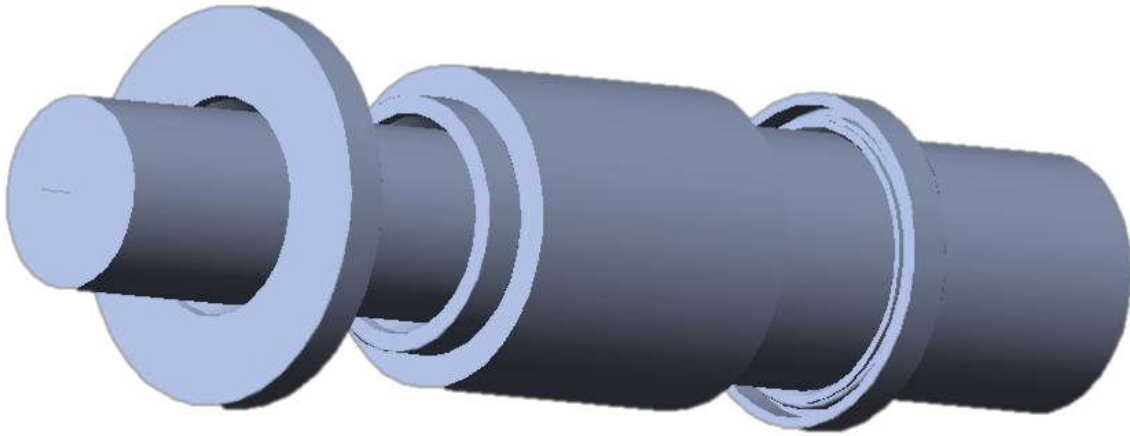
**Solution:-**

In order to insert a 2D Seal on a shaft use these steps:-

1. Firstly, draw the shaft component as shown in Figure.
2. From 2D Shaft Generator dialog box, select the Std.Parts... option, and Shaft Seals.
3. Choose DIN 3760 – A (Left) seal from the standard dialoge box.
4. Specify insertion point on shaft contour: *30,15 Press ENTER.*
5. Direction to [Left/Right]: *R*
6. Drag size: *62 Press ENTER.*
7. Choose DIN 3760 – A (Right) seal from the standard dialoge box.
8. Specify insertion point on shaft contour: *click point on the shaft center where the seals inserted.*
9. Direction to [Left/Right]: *L*
10. Drag size: *40 Press ENTER.*
11. Choose DIN 3760 – A (Right) seal from the standard dialoge box.
12. Specify insertion point on shaft contour: *150,20 Press ENTER.*
13. Direction to [Left/Right]: *R*
14. Drag size: *52 Press ENTER.*

**Exercise:-**

Repeat Example 2DSeal to draw and insert a 3D shaft seal.



## Assembly Drawing

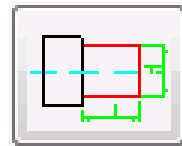
The shaft generator is configured. Now you want to generate the first shaft segments. Verify that the Outer Contour tab is selected.

### To create shaft segments:-

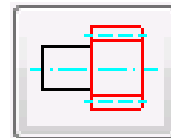
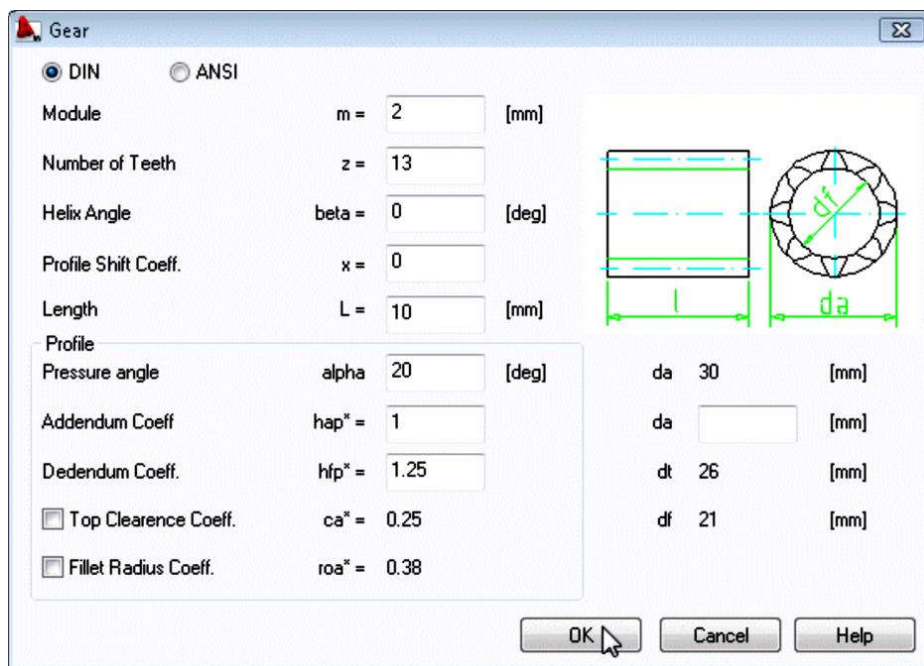
1- Click the lower cylinder button to define a cylinder section, and respond to the prompts as follows:-

Specify length <50>: *Enter 12, press ENTER*

Specify diameter <40>: *Enter 20, press ENTER*



2- Click the gear button, and then enter the values for module, number of teeth, and length as shown in the following figure:

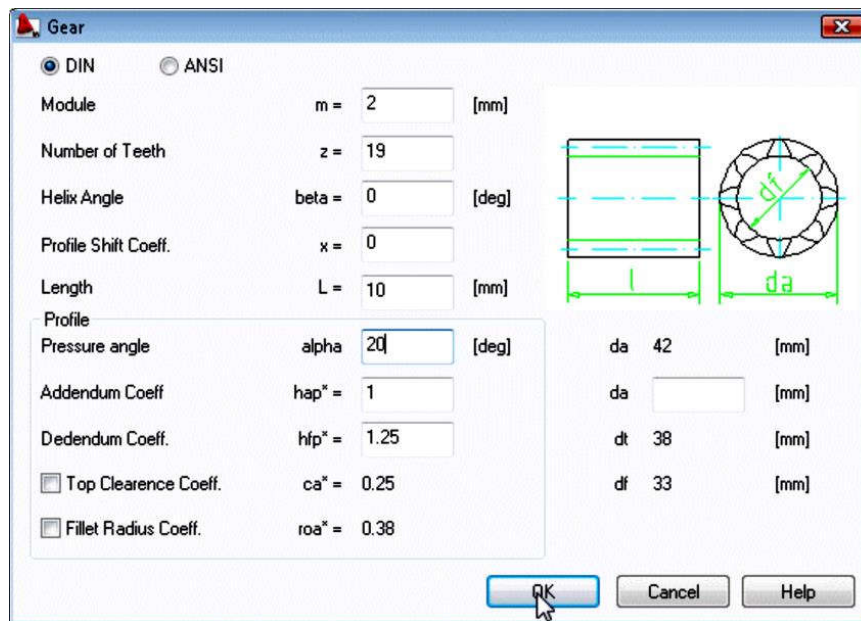


3- Click the lower cylinder button to define a further cylinder section and respond to the prompts as follows:-

Specify length <10>: *Enter 5, press ENTER*

Specify diameter <20>: *Enter 20, press ENTER*

4- Click the gear button, and then enter the values for module, number of teeth, and length as shown in the following figure:-



5- Click the lower cylinder button to define another cylinder section, and then respond to the prompts as follows:-

Specify length <10>: *Enter 4, press ENTER*

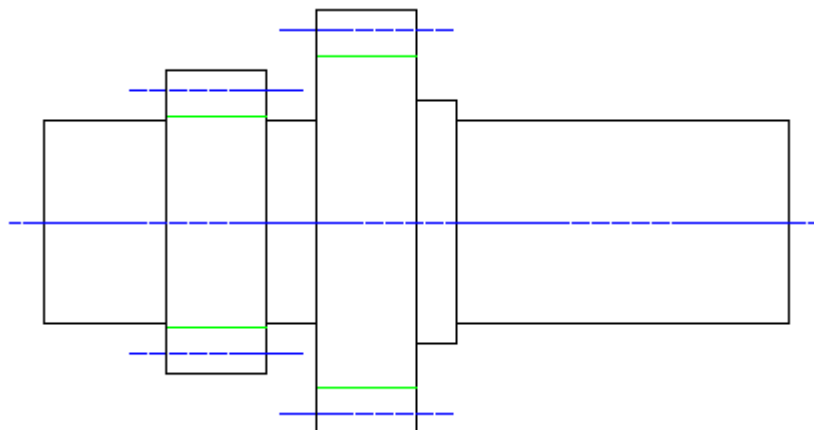
Specify diameter <20>: *Enter 24, press ENTER*

6- Click the lower cylinder button to define another cylinder section, and then respond to the prompts as follows:-

Specify length <4>: *Enter 33, press ENTER*

Specify diameter <24>: *Enter 20, press ENTER*

The first five sections of the shaft are created, as represented in the following figure:-



### **Inserting Chamfers and Fillets:-**

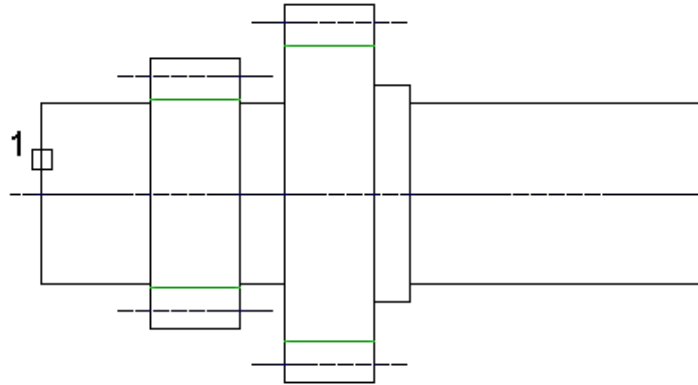
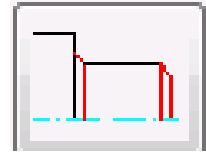
To apply a chamfer and a fillet

7- Click the Chamfer button to apply a chamfer to a shaft section, and then respond to the prompts as follows:-

Select object: *Select the leftmost cylinder section (1)*

Specify length (max. 12) <2.5>: *Enter 2, press ENTER*

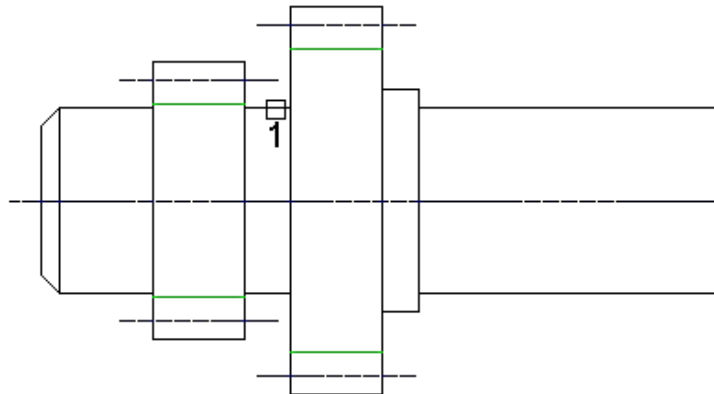
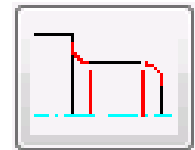
Specify angle (0-79) or [Distance] <45>: *Enter 45, press ENTER*



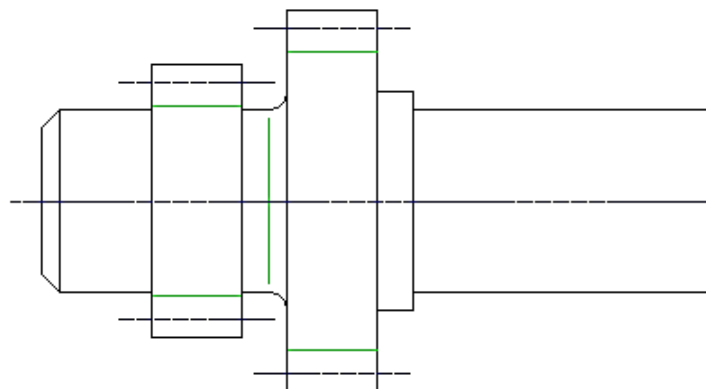
8- Click the Fillet button to apply a fillet to a shaft section, and then respond to the prompts as follows:-

Select object: *Select the cylinder section between the two gears near the second gear(1).*

Enter radius (max. 10.00) <2.50>: *Enter 2, press ENTER*



The shaft looks like the following figure:-



## Editing Shafts and Inserting Sections:-

Edit an existing shaft section and insert a new section. You use the Edit button in the shaft generator to turn on AMPOWEREDIT.

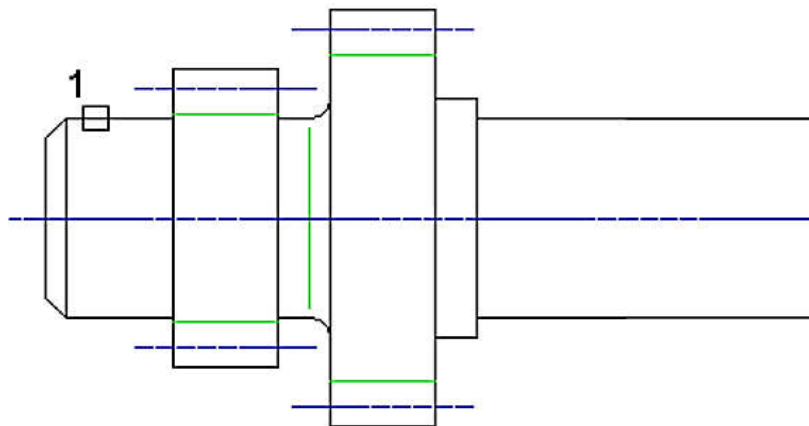
To edit and insert a shaft section

**9-** Click the Edit button, and then respond to the prompts as follows:-

Select object: *Select the first cylindrical section (1)*

Specify length <12>: *Press ENTER*

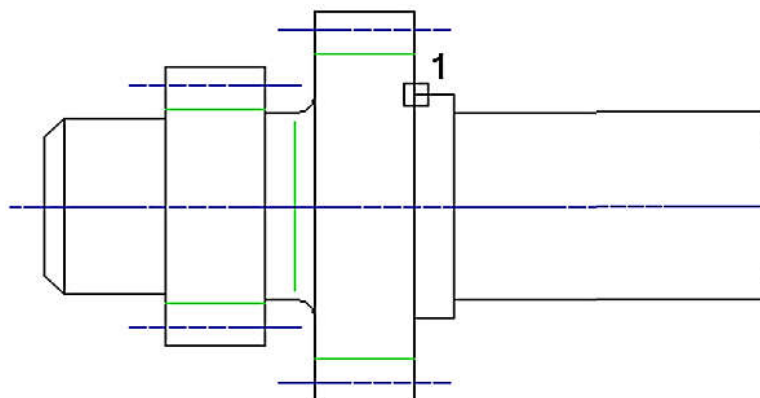
Specify diameter <20>: *Enter 18, press ENTER*



The diameter is changed to 18 while the length remains 12.

**10-** Click the Insert button, and then respond to the prompt as follows:-

Specify point: *Select a point after the second gear (1)*

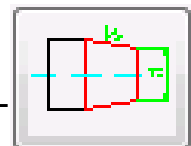


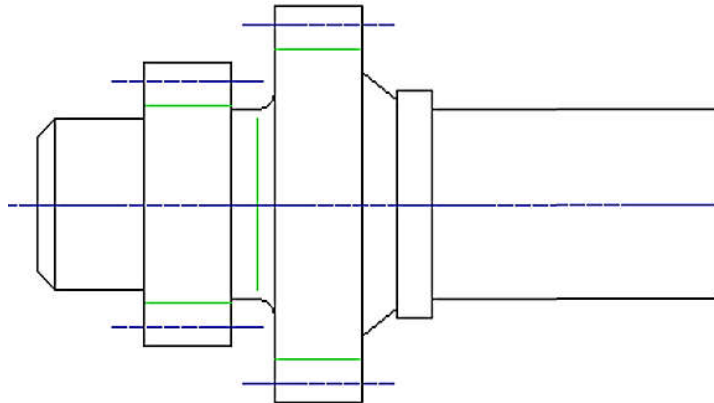
**11-** Click the Slope button, and then respond to the prompts as follows:-

Specify length or [Dialog] <20>: *Enter 4, press ENTER*

Specify diameter at starting point <24>: *Enter 28, press ENTER*

Specify diameter at endpoint or [Slope/Angle] <20>: *Enter 22, press ENTER*





## Replacing Shaft Sections:-

The previously inserted slope needs to be deleted again. To replace a shaft section

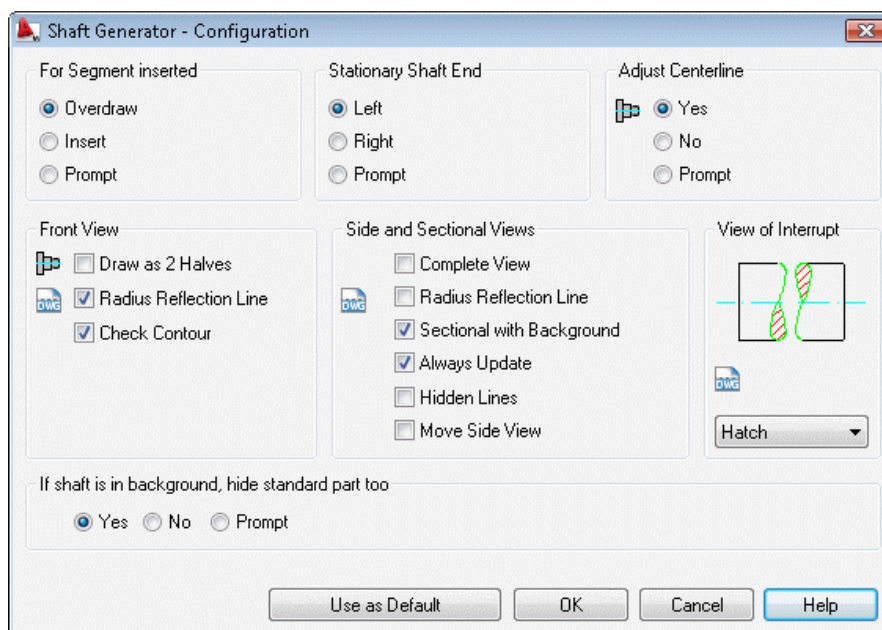
**12-** Click the Undo button.

The previous slope insertion is undone.

Replace an existing shaft section. To do this, change the settings in the configuration.

**13-** Click the Options button to start the shaft generator configuration, and then specify:-

For Segment inserted: **Overdraw**

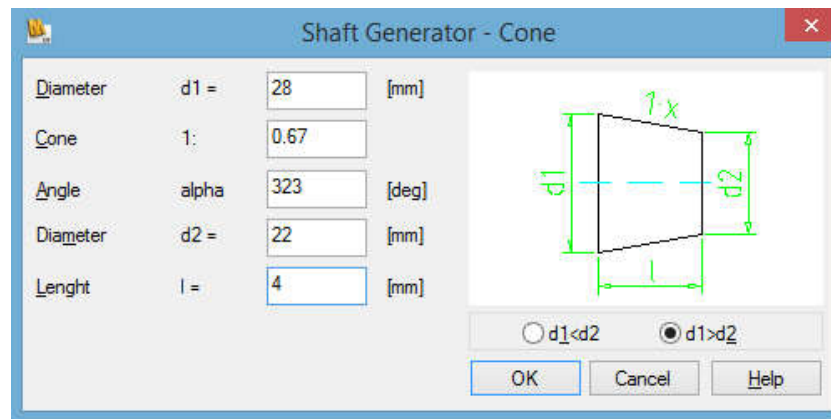


Click OK.

**14-** Click the Slope button, and then respond to the prompt as follows:-

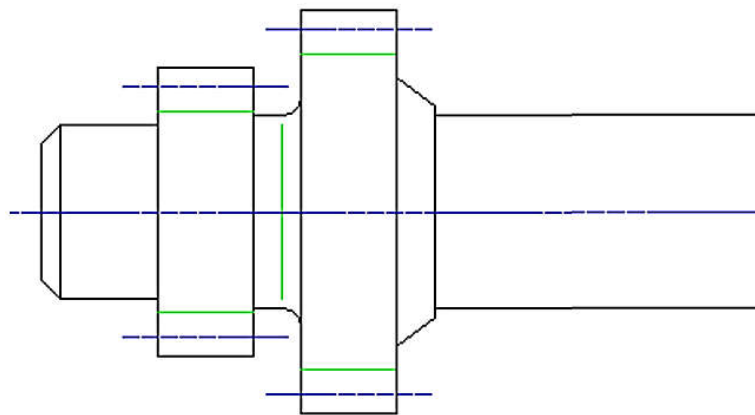
Specify length or [Dialog] <4>: *Enter D, press ENTER*

**15-** In the Shaft Generator - Cone dialog box, specify the following settings.



Click OK.

The slope replaces the cylindrical shaft section



### **Inserting Bearings:-**

To insert a bearing

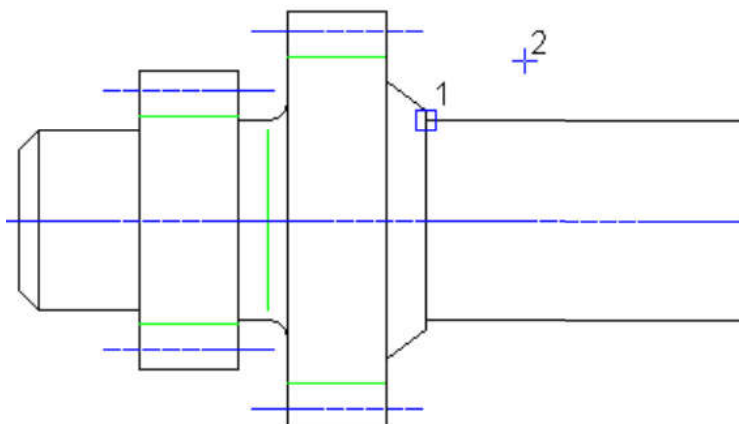
**16-** Click the Standard Parts button, and then select Roller Bearings ➤

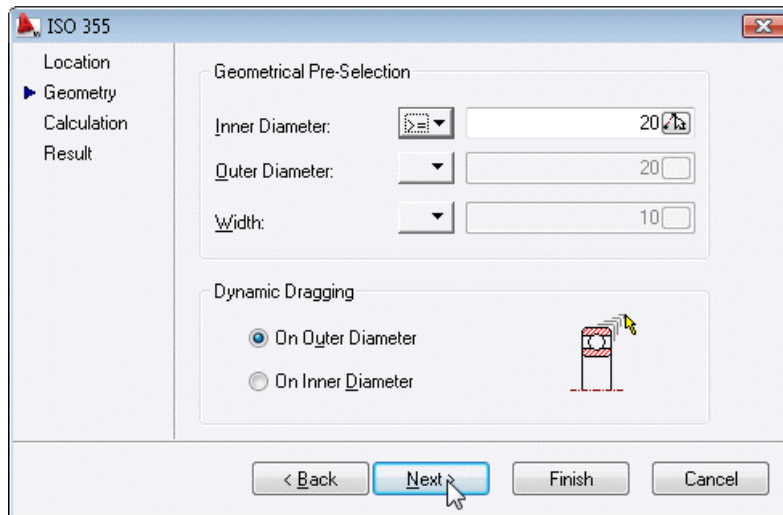
Radial ➤ ISO 355. Respond to the prompts as follows:-

Specify insertion point on shaft contour: *Specify insertion point (1)*

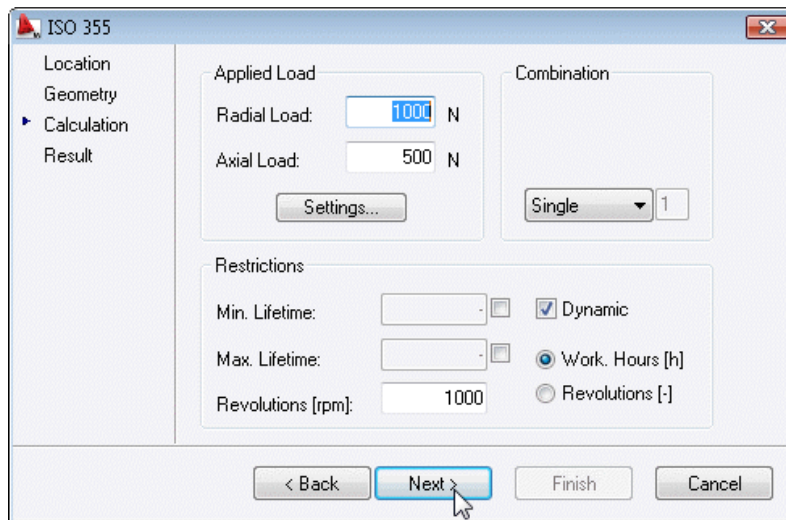
Direction to [Left]: *Select a point to the right (2)*

**17-** In the ISO 355 dialog box, click Next.



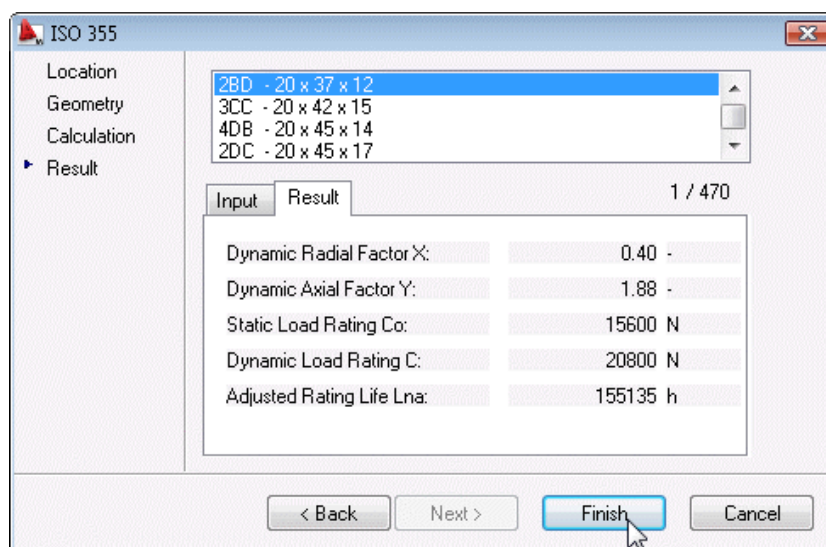


18- Specify the loads, and activate Work Hours as shown in the following.



Click Next.

19- In the ISO 355 dialog box, select the bearing 2BD - 20 x 37 x 12, and then click Finish.

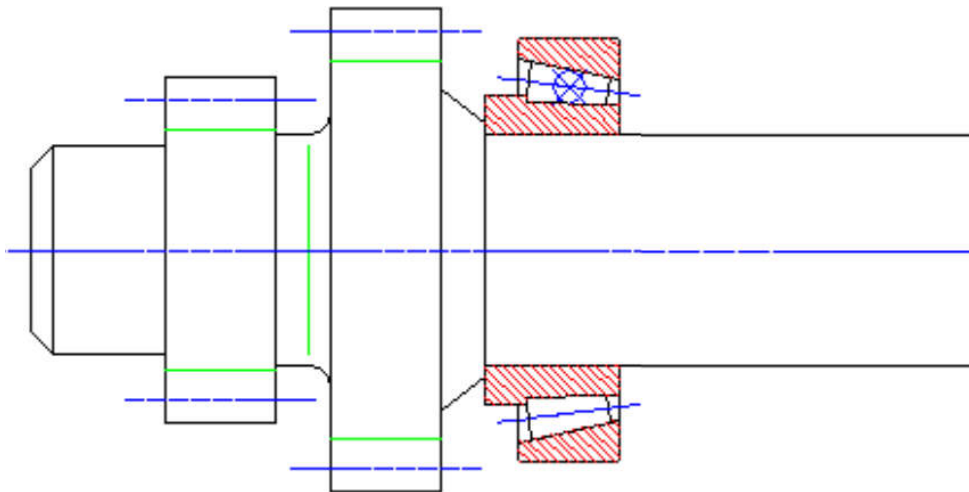


You can drag the cursor to see all available bearing sizes.

**20-** Drag to the size 2BD - 20 x 37 x 12, and then press ENTER.

**21-** In the Create Hide Situation dialog box, click OK.

The bearing is inserted.



**22-** Close the Shaft Generator dialog box.

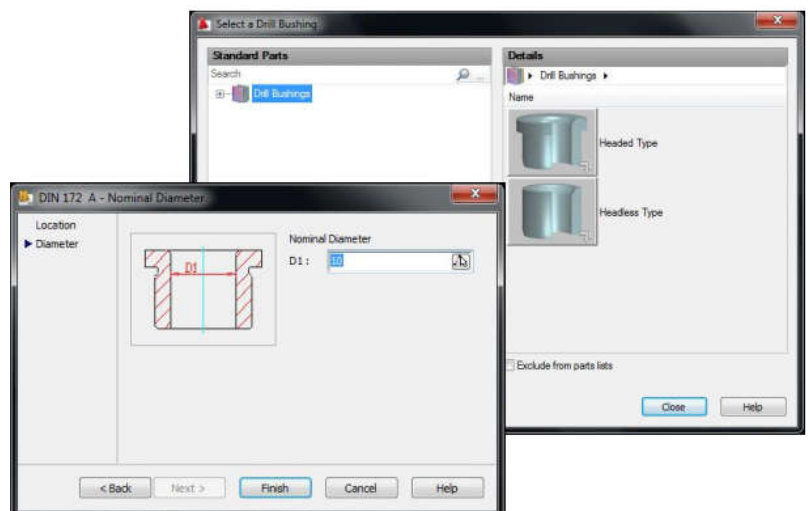
## Drill Bushings Drill Bushings

A **drill bushing** is a tool used in metalworking jigs to guide cutting tools, most commonly drill bits. Other tools that are commonly used in a drill bushing include counter bores, countersinks, and reamers. They are designed to guide, position, and support the cutting tool.



In MDT 2009 program use AMDRBUSH2D to insert drill bushings from the installed standard parts **OR** Content2D → Drill Bushings → Drill Bushings

1. Select the Bushing type.
2. Select the suitable standard of the type.
3. Select the projection view.
4. Specify insertion point.
5. Specify rotation angle <0>:
6. Enter the value of inner diameter & press finish.
7. Drag size: Specify the length using the pointing device, or enter a value.



❖ Use AMDRBUSH3D to insert drill bushings from the installed standard parts  
**OR** Content3D → Drill Bushings → Drill Bushings

### **COMMAND LINE OPTIONS**

Select first point [Concentric/cylinder/two Edges]: *Specify the first point or select the option you want to use to insert the 3D part.*

Select second point [Concentric/cylinder/two Edges]: *Select the option you want to use to insert the 3D part.*

Drag size [Dialog]: *Drag the size by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table.*

### **COMMAND LINE DEFINITIONS**

#### **Concentric**

Select circular or elliptical edge: *Click the edge of the area where you want to insert the standard part. Choose insertion direction [Flip/Accept] <Accept>: Press ENTER. In the Nominal Diameter dialog box, select the diameter for the standard part.*

Drag size [Dialog]: *Drag the size by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table.*

#### **Cylinder**

Select cylindrical face: *Click the edge of the area where you want to insert the standard part. Select location on face: Specify the location by moving the cursor.*

Choose insertion direction [Flip/Accept] <Accept>: *Press ENTER. In the Nominal Diameter dialog box, select the diameter for the standard part.*

Drag size [Dialog]: *Drag the size of the standard part by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table .*

#### **Two Edges**

Select first edge or planar face: *Select the first edge.*

Select second edge or planar face: *Select the second edge.*

Select work plane or planar face: *Select a work plane.*

Specify location: *Move the cursor to specify the insertion point.*

*Enter distance from first geometry (highlighted).*

[Associate to/Equation assistant] <value>: *Press ENTER.*

*Enter distance from second geometry (highlighted)*

[Associate to/Equation assistant] <value>: *Press ENTER.*

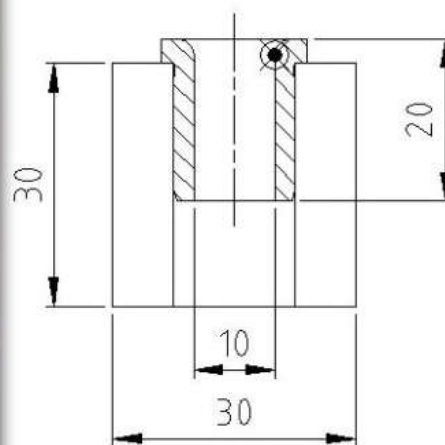
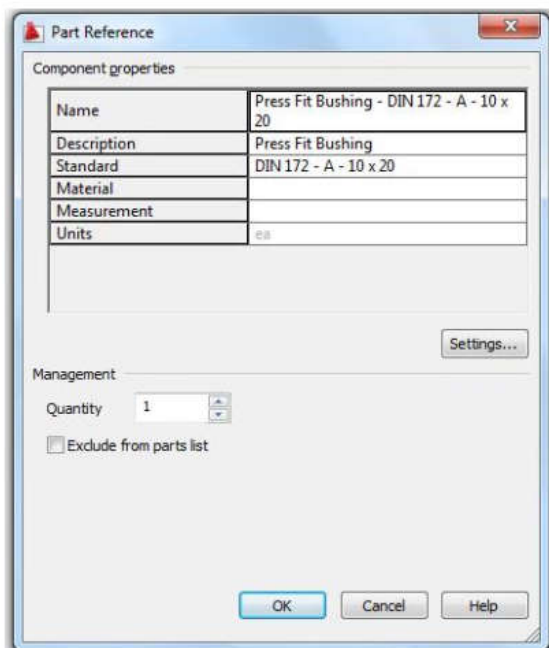
Choose insertion direction [Flip/Accept] <Accept>: *Press ENTER. In the Nominal Diameter dialog box, select the diameter for the standard part.*

Drag size [Dialog]: *Drag the size of the part by moving the cursor in the insertion direction. If you select Dialog, you choose the part from a table.*

❖ Also you can insert drill bushings with holes from the installed standard parts by writing AMDRBUSHHOLE2D **OR** AMDRBUSHHOLE3D in the command line **OR** from Content2D/3D → Drill Bushings → Drill Bushings with holes.

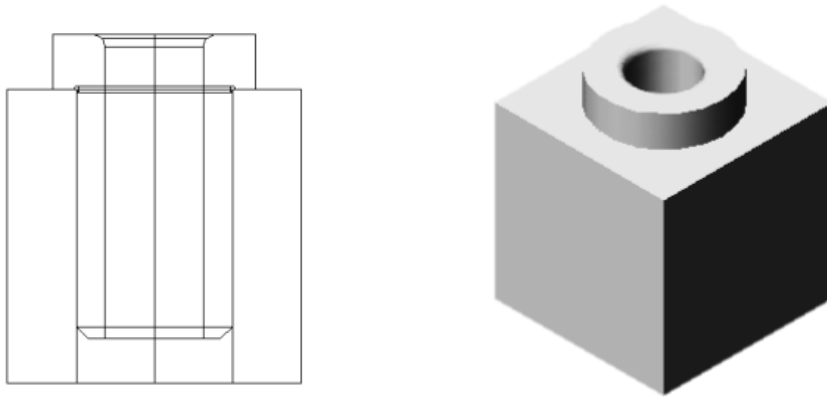
### **Exercise DRBUSH1:-**

Draw a rectangle with dimensions of (30 ×30) and insert a Drill Bushing as appear in Figure below.

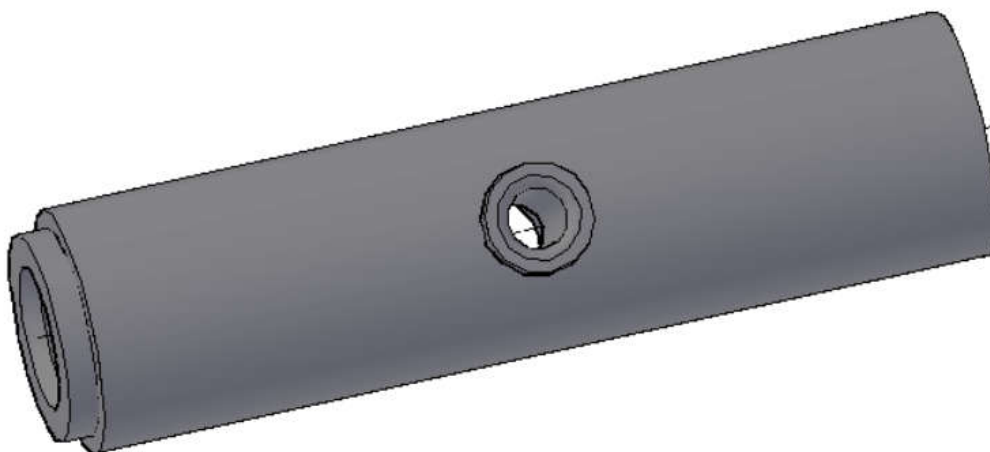
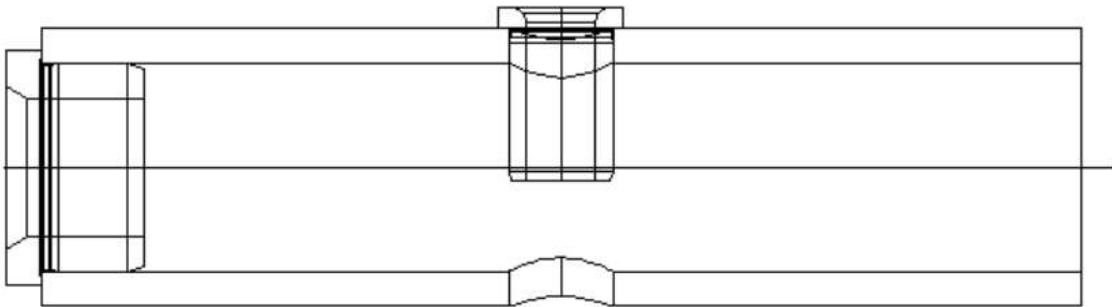


**Exercise DRBUSH2:-**

Draw a Box with dimensions of (30 × 30 × 30) and insert a 3D Drill Bushing with hole as appear in Figure below. ( $D_1 = 10$  and Drag size = 25)

**Exercise DRBUSH3:-**

Draw a Shaft with a Drill Bushing with holes. (**Note:-** choose the dimensions by yourself).



## Springs

There are four types of springs can be drawn in MDT 2009 program:-

- 1- Compression spring:-** A spring type that can be compressed and can absorb pressure forces.
- 2- Extension spring:-** A spring type that can absorb tension forces.
- 3- Torsion spring:-** A spring type that can absorb torque forces.
- 4- Belleville spring washer:-** A washer-type spring that can sustain relatively large loads with small deflections. The loads and deflections can be increased by stacking the springs.

### 1- Compression spring:-

To insert a compression spring (2D)

- a) On the command line, enter AMCOMP2D.
- b) In the Select Compression Spring dialog box, select a Catalog or Standard.
- c) Select Front View or Sectional View.
- d) Click to specify the starting point.
- e) Click to specify the direction.
- f) In the Compression Springs dialog box, from the Specification dropdown list, select the insertion method. There are four standard cases, and one detailed case.
- g) Add restrictions and choose Next.
- h) If you selected Modified Design in the Select a Compression Spring dialog box in step 2, select the material and wire diameter and choose Next.
- i) Select one or more springs from the table.
- j) Choose Finish.
- k) Specify the length of the spring by dragging and clicking.
- l) Select an object in the drawing to be inside the spring (hidden by the spring) or press ENTER.
- m) Press ENTER. (The compression spring is inserted)

In MDT 2009 program Use AMCOMP3D to insert and calculate compression springs OR from (Menu ⇒ Content 3D ⇒ Springs ⇒ Compression). In order to make the calculation as simple and as clear as possible, the

spring types are inserted using the same methods. You use dialog boxes for the selection, calculation, and insertion process. Use the Select Compression Spring dialog box to select the standard, the view spring type and select options.

## COMMAND LINE OPTIONS

Specify starting point:

Specify the insertion point for the spring.

Specify point in direction of spring axis:

Specify the second point for the direction of the spring. In the graphic area click in direction of the spring. In the Compression Springs dialog box, select other options.

## DIALOG BOX DEFINITIONS

The spring insertion and calculation is controlled by a sequence of dialog boxes depending upon the spring insertion method you selected.

### *Select Compression Spring dialog box*

Use the Select Compression Spring dialog box to select the spring insertion method or to insert an empty form.

### *Standards*

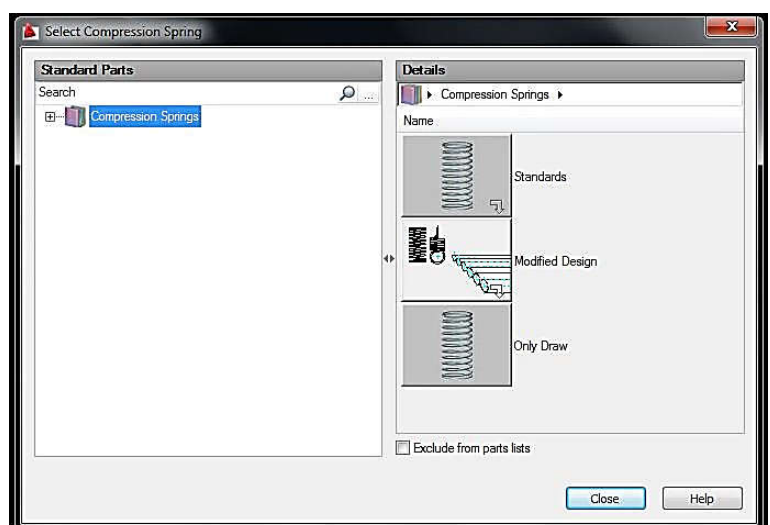
Lets you select, draw and calculate a spring from a Standard, or Manufacturer table.

### *Modified Design*

Lets you select, draw and calculate a specially produced spring.

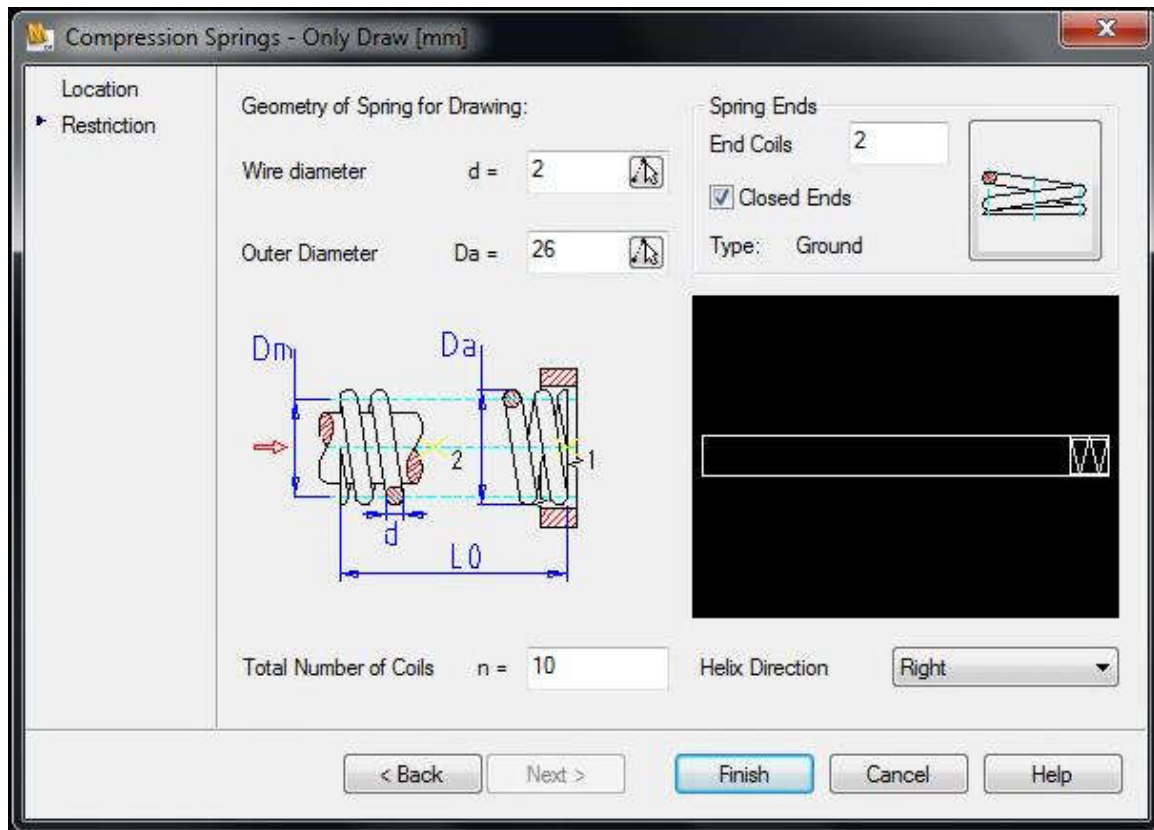
### *Only Draw*

Lets you draw a spring without calculation.



## Compression Springs - Only Draw dialog box

Use the Drawing dialog box to define the geometry of the spring in the drawing



Wire diameter  $d$

Enter a spring wire diameter or click on the measure button and determine this value from the graphic area.

$D_a$

Specifies the Outer Diameter of the spring. Enter a value or click on the measure button and determine this value from the graphic area.

$n$

Specifies the total Number of Coils of the spring including the ends of coils. Enter a value.

**Helix Direction**

Select one of two spring helix directions: Right, or Left.

### ***Spring Ends***

#### ***End Coils***

Enter a total number of end coils for both spring ends.

#### **Closed Ends**

Use this check box to define closed or open ends of spring.

#### ***Spring end shapes button***

From the Type of Ends dialog box, select the required spring end type.

There are three types of compression spring ends: Ground, Round and Forged.

### **Compression Springs - Select from tables dialog box**

#### ***Default Restrictions***

#### ***Specification***

Specifies a combination of force and geometrical restrictions: Load,

Deflection / Load, Length / 2 Load, 2 Length / 2 Loads, Travel / Detailed.

Select an appropriate combination based on the disposable input data. After selecting the specification, a spring picture with dimension buttons and a value table of restrictions appears.

#### ***Absolute set***

Contains two options: Lengths and Forces. If the Force is of the absolute value, then the length is within the range from - to, and the other way around.

#### ***Restrictions table***

To insert a restriction into the table, click on the dimension button (for example: Da) in the spring picture to the left of the table. A new empty row appears in the table. To enter values into the cells, click on a cell. You can also enter geometrical values (length and diameter) using the measure button in the cell and then making selections in the graphic area. Use the buttons in the cells (< >, < =, = >, =) to add further restrictions to the spring. To activate a dynamically loaded spring use the Dynamic Loads check box. This option affects calculation of stresses.

#### ***Settings***

Opens the Additional Settings dialog box.

***Additional Calculation Settings***

Opens the Compression Springs - Additional Calculation dialog box.

***Back***

Finds appropriate springs in the table according to restrictions and displays them in the result table along with important dimensions and calculation results.

***Next***

Lets you change the location and direction of the spring.

***Finish***

Finds and inserts the appropriate springs in the table into your drawing. You can dynamically drag the springs in the drawing.

***Select all toolbar***

Lets you quickly select all springs in the result list. If you select more than one spring from the result list, dynamic dragging is activated when you press the Finish button so you can select more springs from the graphic area

***Results: Loads tab******Spring dimensions:***

d-wire diameter x Da-outer diameter x L0-unloaded length. Depending on the selected standard, instead of outer diameter Da mean diameter Dm can be displayed.

***Force:***

F1- load in state 1, F2- load in state 2, Fn- maximum allowed load.

***Deflection***

s1- deflection in state 1, s2- deflection in state 2, sn- maximum allowed deflection.

***Length:***

lengths of spring caused by corresponding forces (L1, L2, Ln). Number of displayed forces depends on selected specification

**Results: Stresses tab****Stress:**

stress in spring calculated according to the forces.

**Allowed:**

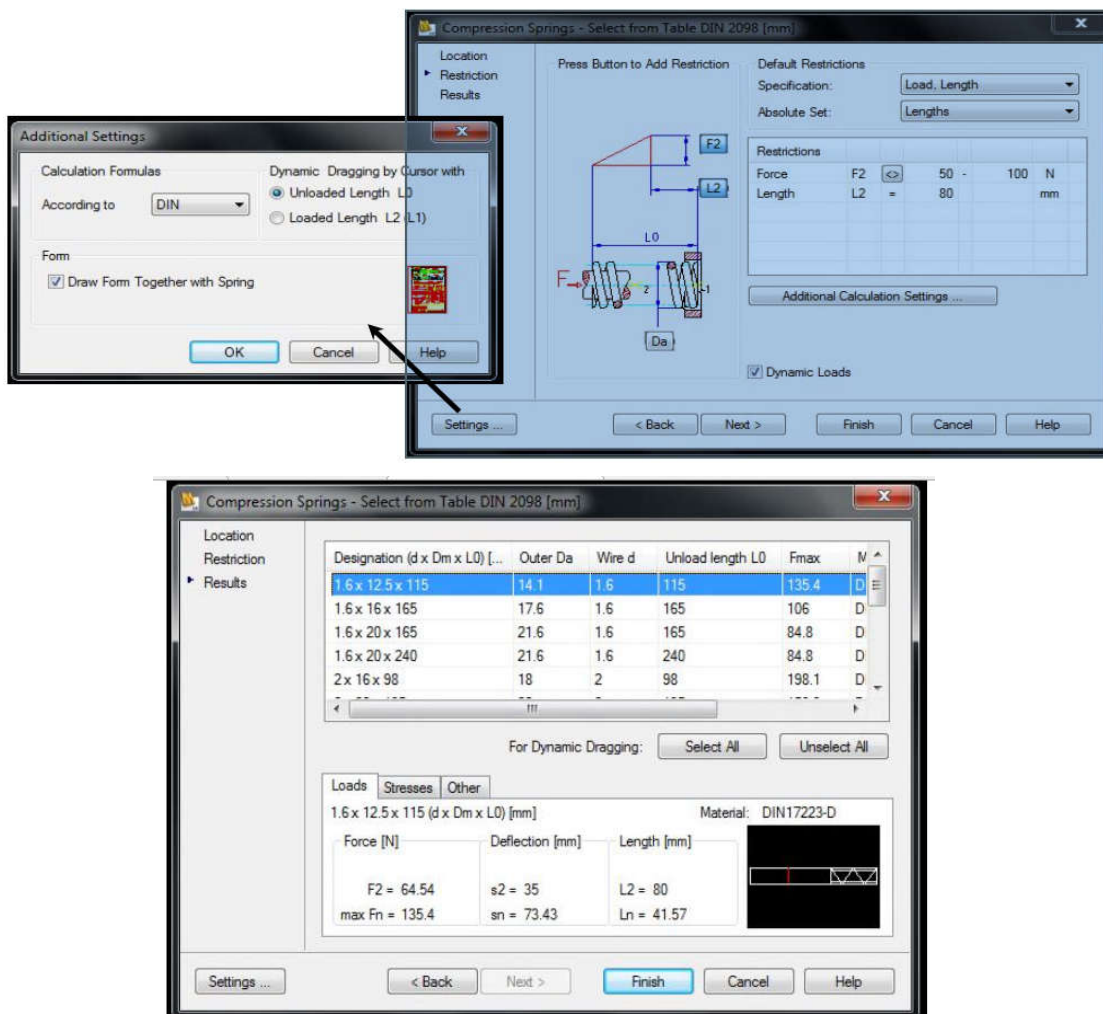
allowed stress in spring

**Safety:**

safeties calculated as a ratio between stress and allowed stress in the spring

**Results: Other tab**

Displays information about the spring such as natural frequency of spring and allowed number of stress cycles. If you defined inputs in the Additional calculation settings dialog box then this tab displays information about the spring's safety against buckling and against influence of shear load.



## **Compression Springs - Modified design dialog box**

### ***Material***

Specifies required material in accordance with the selected standard

### ***Wire diameter $d$***

Specifies wire diameters in accordance with the selected standard.

### ***Outer Diameter $D_o$***

Enter a value for either the Outer Diameter  $D_o$ , or index ratio  $D_m/d$ , where  $D_m$  is mean coil diameter,  $d$  is wire diameter of the spring.

### ***Unloaded Length $L_0$***

Specifies the unloaded length of the spring. If the spring is specified by the force and deflection, the minimum permissible length  $L_0$  is calculated and displayed. If the spring is specified by the force and loaded length, the maximum permissible length  $L_0$  is calculated. If the spring is specified by 2 forces and 2 lengths there is only one possible length  $L_0$ , which is automatically calculated.

## **Spring Ends**

### ***End Coils***

Enter a required number of end coils together for both spring ends.

### ***Closed Ends***

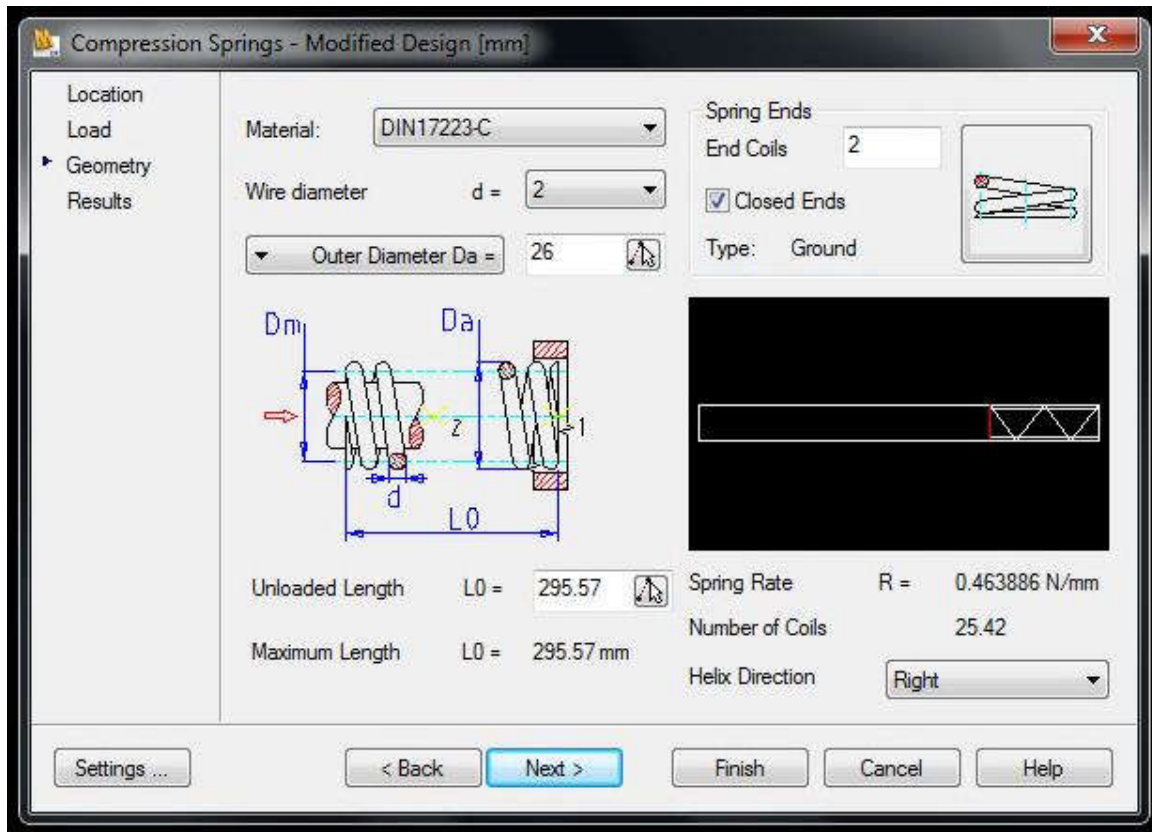
Use this check box to define closed or open ends of spring.

### ***Spring end shapes button***

Click on the spring end picture and from the Type of Ends dialog box select the required spring end type. There are three types of compression spring ends: Ground, Round and Forged.

### ***Helix Direction***

Select one of two spring helix directions: Right, or Left.



**Example** Comp2D:- create a compression spring in two different compression situations. You calculate and insert the springs in the existing drawing.

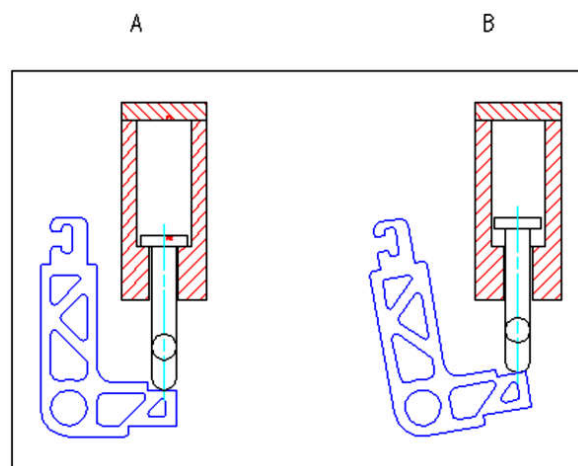
**Solution:-**

Open a drawing

1. Open the file tut\_spring.dwg in the tutorials folder at:

Windows Vista™:C:\Users\Public\PublicDocuments\Autodesk\ACADM2009\Acadm\Tutorial

OR Windows®XP:C:\DocumentsandSettings\AllUsers\SharedDocuments\Autodesk\ACADM2009\Acadm\Tutorial



## 2. To specify a spring

- Start the Compression Spring command. On the command line, enter AMCOMP2D.
- In the Select Compression Spring dialog box, click Standards > SPEC® Catalog A > Front View.

## 3. Respond to the prompts as follows:

Specify starting point: Specify the starting point (1)

Specify direction: Specify endpoint (2)

## 4. To specify the spring restrictions

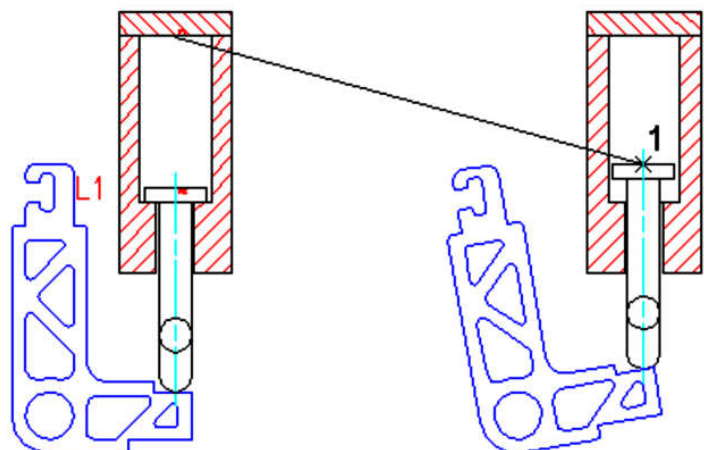
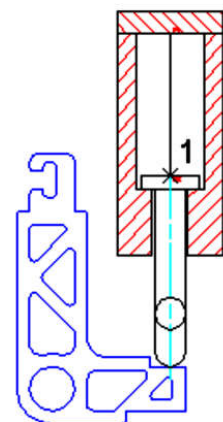
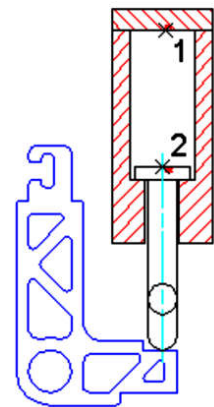
- In the Compression Springs - Select from Table SPEC® Catalog A [mm] dialog box, specify:
  - Specification: 2 Loads, 2 Lengths
  - Absolute Set: Lengths
  - Click the **Da** button. Enter the value 15.

5. In the Compression Springs - Select from Tables SPEC® Catalog A [mm] dialog box, click the value field for the length L1, and then choose the pick icon.

6. Respond to the prompts as follows:  
Specify point for spring length L1: Select a point on the spring pressure plate (1).

In the Compression Springs dialog box, click the value field for the length L2, and then choose the pick icon.

7. Specify point for spring length L2: Select a point on the spring pressure plate in view B (1).



8. Then choose Select All to select all possible springs for the dynamic dragging

9. To insert a spring

Drag the cursor until the tooltip reads SPEC - 1.6 x 14.1 x 36, and then click.

Respond to the prompts as follows: Topical Length (14.28 - 36) [Force/Deflection] <32.01>: Select a point on the spring pressure plate (1).

10. Continue to respond to the prompts as follows: Select rod (only closed contours) <Enter=continue>:

Press ENTER

The spring is inserted as shown below.

11. To create a view of a spring with Power View

Start the Power View command. On the command line, enter AMPOWERVIEW.

Respond to the prompts as follows: Select objects: Select the spring in view A

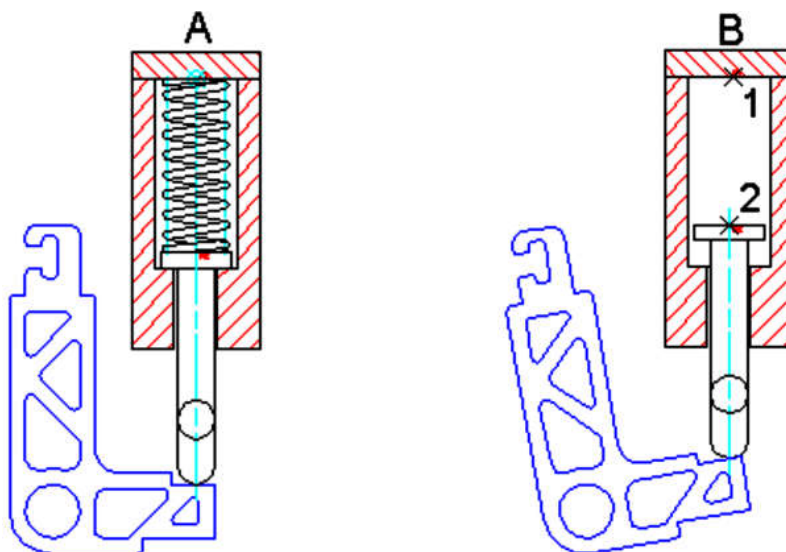
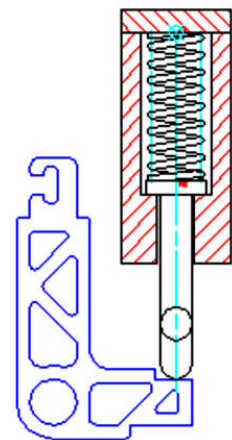
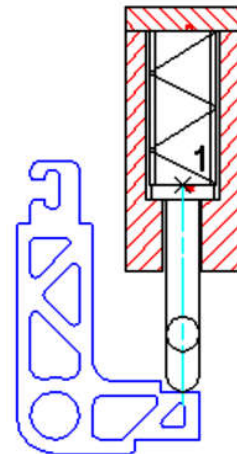
In the Select New View dialog box, select Front View.

Specify starting point: Select point (1) in view B

Specify direction: Select point (2) in view B

Topical Length (14.28 - 36)[Force/Deflection]<32.01>: Select the lower contact point of the compressed spring

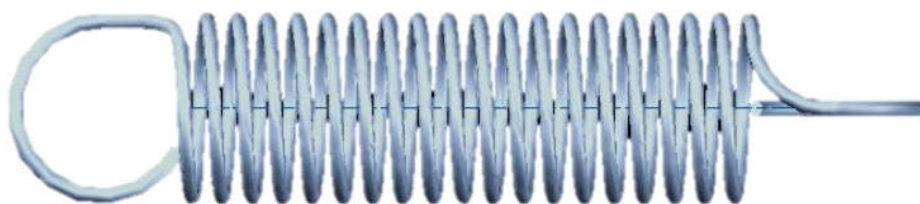
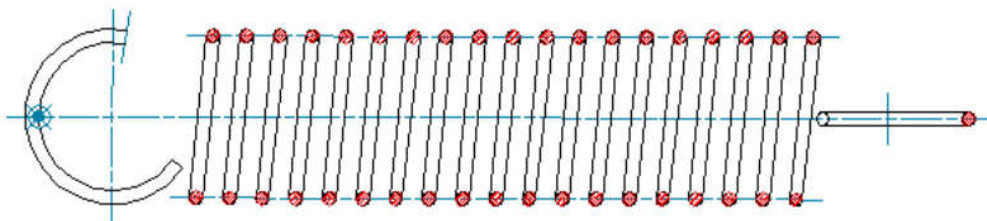
Select rod (only closed contours) <Enter=continue>: Press ENTER



## 2- Extension spring:-

To insert an extension spring (2D)

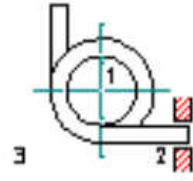
- a) On the command line, enter AMEXT2D.
- b) In the Select an Extension Spring dialog box, select a catalog or standard.
- c) Choose Front View or Sectional View.
- d) Click to specify the starting point.
- e) Click to specify the direction.
- f) In the Extension Springs dialog box, from the Specification drop-down list, select the insertion method. There are four standard cases, and one detailed case.
- g) Add restrictions and choose Next.
- h) If you selected Modified Design in the Select an Extension Spring dialog box in step 2, select the material and wire diameter and choose Next.
- i) Select one or more springs from the table.
- j) Choose Finish.
- k) Specify the length of the spring by dragging and clicking. The Extension Spring- Angle of View for Drawing dialog box appears.
- l) Select the angle of view for drawing.
- m) Choose Next.
- n) Select an object in the drawing to be inside the spring (hidden by the spring) or press ENTER.
- o) Press ENTER. (The extension spring is inserted).



### 3- Torsion Spring:-

To insert a torsion spring (2D)

- a) On the command line, enter AMTOR2D.
- b) In the Select a Torsion Spring dialog box, select a catalog or standard.
- c) Specify front, side or sectional view.
- d) In the Type of Ends of Torsion Springs, choose a type of ends and choose OK.
- e) Click to specify a center point.
- f) Click to specify a support point of the fixed end.
- g) Click to specify the direction of torsion.
- h) If you selected Modified Design in the Select a Torsion Spring dialog box in step 2, specify the lever.
- i) In the Torsion Spring dialog box, from the Specification drop-down list, select the insertion method. There are four standard cases, and one detailed case.
- j) Choose Next.
- k) If you selected Modified Design in the Select a Torsion Spring dialog box in step 2, specify the material and wire diameter of the spring.
- l) Choose Next.
- m) Select one or more springs from the table.
- n) Choose Finish.
- o) Specify the helix direction and the current angle.
- p) If you selected a side or sectional view, select an object in the drawing to be inside the spring (hidden by the spring) or press ENTER.
- q) The torsion spring is inserted.



Use AMTOR3D to insert and calculate torsion springs:-

In order to make the calculation as simple and as clear as possible, the spring types are inserted using the same methods. You use dialog boxes for the selection, calculation, and insertion process. Use the Select Torsion Spring dialog box to select the standard, the view spring type and select options.

#### COMMAND LINE OPTIONS

The points you select when creating 3D torsion spring are defined in two

perpendicular planes. The spring axis is defined in one plane (for example XY) and the spring ends are defined in a perpendicular plane (for example YZ).

help you identify points, set the isometric view (View → 3D Views → Isometric).

Specify starting point: Specify first point of spring axis.

Specify point in direction of spring axis [Flip]: Specify a second point for the direction of the spring axis.

Specify support point of fixed end [Axis]: Specify the support point of the fixed end: fixed or movable. Enter A to select the Axis option, which lets you rotate the coordinate system.

Specify direction of torsion: Specify the direction of torsion, clockwise or counterclockwise.

Specify lever: Specify radius of lever, the distance between the centerpoint of the spring and the point of force acting on the movable end.

Topical Length (23.36 - 260) [Force/Deflection] <23.36>: Enter a value for the angle between the ends of the spring, or draw it on the screen. The angle must be between the maximal loaded angle and the unloaded angle. Use the Force option to define the angle in terms of the corresponding value of force between the minimum and maximum load.

#### DIALOG BOX DEFINITIONS

The spring insertion and calculation is controlled by a sequence of dialog boxes depending upon the spring insertion method you selected.

Select a Torsion Spring dialog box

Use the Select Torsion Spring dialog box to select the spring insertion method or to insert an empty form.

Torsion Springs - Only Draw dialog box

Use the Drawing dialog box to define the geometry of the spring in the drawing.

Wire diameter d

Enter a spring wire diameter or click on the measure button and determine this value from the graphic area.

Outer Diameter Da

Specifies the Outer Diameter of the spring. Enter a value, or click on the

measure button and determine this value from the graphic area.

(n) Specifies the total Number of Coils of the spring including the ends of coils. Enter a value.

Helix Direction

Select one of two spring helix directions: Right, or Left.

Ends of Spring

Enter lengths of spring arms.

Torsion Springs - Type of Ends of Torsion Springs dialog box

Select an option to specify the type of fixed and movable ends: tangential or bent ends.

Torsion Springs - Select from Tables dialog box

Default Restrictions

Specification

Specifies a combination of force and geometrical restrictions: Load, Deflection / Load, Angle between Ends / 2 Load, 2 Angles / 2 Loads, Travel / Detailed. Select an appropriate combination based on the disposable input data. After selecting the specification, a spring picture with dimension buttons and a value table of restrictions appears.

Absolute set

Contains two options: Angles and Loads. If the Load is of the absolute value, then the Angle is within the range from - to, and the other way around.

Loads

Select how you want to define loads - using moments or forces

Restrictions table

To insert a restriction into the table, click on the dimension button (for example: Dd) in the spring picture to the right of the table. A new empty row appears in the table. To enter values into the cells, click on a cell. You can also enter geometrical values (angle and diameter) using the measure button in the cell and then making selections in the graphic area. Use the buttons in the cells (<>, <=, =>, =) to add further restrictions to the spring. To activate a dynamically loaded spring use the Dynamic Loads check box. This option affects calculation of stresses.

Settings

Opens the Additional Settings dialog box.

Lever

Lets you edit the radius of lever, which is the distance between the centerpoint of the spring and the point of force acting on the movable end. In the front view, by default, you can pick the value in the graphic area.

Fixed

Lets you edit the radius of the fixed end, which is the distance between the centerpoint of the spring and the support point on the fixed end. In the front view, by default, you can pick the value in the graphic area.

Bend of Ends

Opens the Select type of bending ends dialog box. Lets you define if a bend is possible for spring ends.

Back

Finds appropriate springs in the table according to restrictions and displays them in the result table along with important dimensions and calculation results.

Next

Lets you change the location and direction of the spring.

Finish

Finds and inserts the appropriate springs in the table into your drawing. You can dynamically drag the springs in the drawing.

Select all toolbar

Lets you quickly select all springs in the result list. If you select more than one spring from the result list, dynamic dragging is activated when you press the Finish button so you can select more springs from the graphic area

Results: Loads tab

Spring dimensions:

d- wire diameter x Di- inner diameter x Dd- mandrel diameter. Depending on the selected standard, the unloaded angle between spring ends can be displayed instead of the inner diameter Di.

Force:

F1- load in state 1, F2- load in state 2, Fn- maximum allowed load.

Deflection:

a1- deflection in state 1, a2- deflection in state 2, an- maximum allowed

deflection.

Angles between ends:

Angles caused by corresponding forces ( $b_1$ ,  $b_2$ ,  $b_n$ ). Number of displayed forces depends on selected specification.

Results: Stresses tab

Stress:

Stress in spring calculated according to the forces.

Allowed:

Allowed stress in spring.

Safety:

Safeties calculated as a ratio between stress and allowed stress in the spring.

Results: Other tab

Displays information about the spring such as allowed number of stress cycles.

Torsion Springs - Angle of View for Drawing dialog box

This dialog box appears only if you selected side or sectional view.

Angle of View for Drawing

Specify the angle of view on the spring. The angle is represented by a yellow arrow in the dynamic picture in the dialog box

Angle of Spring Ends

Specify the angle between the ends of the spring. The angle must be defined in degrees between the maximal loaded angle and the unloaded angle. You can also specify the corresponding force or angle of deflection instead of the angle between the ends.

Torsion Springs - Additional Settings dialog box

Calculation Formulas

There are two possible calculations: according to DIN or ANSI formulas.

Type of calculation is set to default according to selected catalog, but can be changed. It is possible to change this input only in the Location or Restriction subdialog.

Dynamic Dragging by Cursor with

If you specify dynamic dragging with loaded angle, a spring with a loaded

angle indicated by a red line, closest to the position of the cursor is chosen. If you specify dynamic dragging with outer diameter, a spring with an outer diameter, closest to the position of the cursor is chosen.

Draw form together with spring

After activating this check box the form will be placed together with inserted spring into the drawing.

Torsion Springs - Modified Design dialog box

Material

Specifies required material in accordance with the selected standard

Wire diameter  $d$

Specifies wire diameters in accordance with the selected standard.

Outer Diameter  $D_a$

Enter a value for either the Outer Diameter  $D_a$ , or index ratio  $D_m/d$ , where  $D_m$  is mean coil diameter,  $d$  is wire diameter of the spring.

Overlapping Coils

If you check this check box, the coils are close to each other and the unloaded length  $L_k$  is defined by the number of coils and the wire diameter.

Helix Direction

Select one of two spring helix directions: Right, or Left.

Unloaded Length  $L_0$

Specifies the unloaded length of the spring.

Angle of Ends

Lets you change the angle between the ends of the spring. Angle of ends, wire diameter and outer diameter of the spring are interdependent. If you change one, the others are automatically recalculated.

Ends of Spring

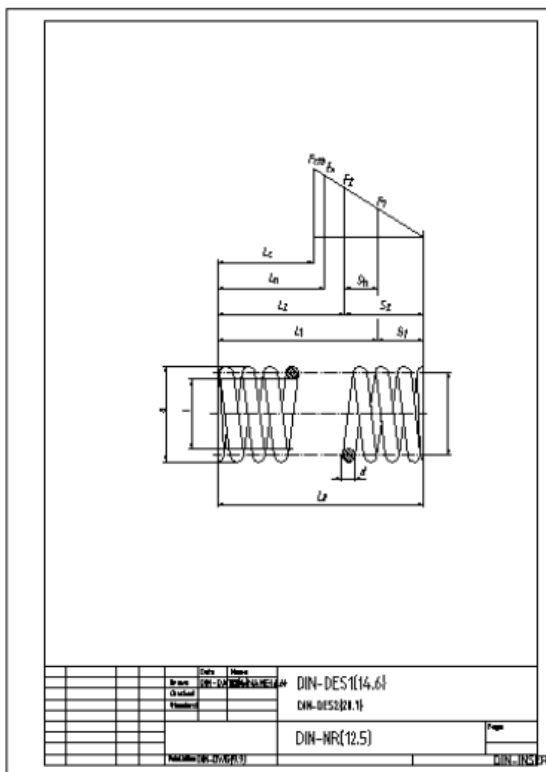
Enter the lengths of the spring arms.

### ❖ **To Insert an Empty Form for Springs:-**

1. On the command line, enter a command as appear in the table below

Type of spring	Command
Compression	AMCOMP2D
Extension	AMEXT2D
Torsion	AMTOR2D
Belleville	AMBELL2D

- In the respective dialog box, select empty form.
- Choose OK.
- Specify an insertion point.
- Double-click the form.
- In the Edit Attributes dialog box, enter the values you need.
- Choose OK.



d	SPRING-DIA.01 (mm)	F <sub>1</sub>	SPRING-SUBRING-RM(14)	S <sub>1</sub>	SPRING-HUB.01 (mm)
	SPRING-DI(3.0) (mm)	L <sub>1</sub>	SPRING-L.1(8.0) (mm)	S <sub>2</sub>	SPRING-TURN(8.0) (N/mm <sup>2</sup> )
r	SPRING-GA(8.0) (mm)	T <sub>1</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	H	SPRING-VARIABLE
r	SPRING-GA(8.0) (mm)	T <sub>2</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	N	SPRING-RC(2.0) (mm)
L	SPRING-GA(8.0) (mm)	F <sub>2</sub>	SPRING-SPRING-RM(14)	F	SPRING-RC(2.0) (mm)
n	SPRING-AC(2.0) (mm)	L <sub>2</sub>	SPRING-L.1(8.0) (mm)	F <sub>1</sub>	SPRING-HUB.01 (mm)
n <sub>1</sub>	SPRING-TC(3.0) (mm)	T <sub>1</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	R <sub>1</sub>	SPRING-CB(3.0) (N/mm <sup>2</sup> )
L <sub>1</sub>	SPRING-L.1(8.0) (mm)	T <sub>2</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	T	SPRING-HUB.01 (mm)
L <sub>2</sub>	SPRING-FIL(3.0) (mm)	F <sub>1</sub>	SPRING-FIL(3.0) (mm)	??	SPRING-IMP(1.0) (mm)
T <sub>1</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	L <sub>1</sub>	SPRING-L.1(8.0) (mm)		
T <sub>2</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	T <sub>1</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )		
T <sub>2</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )	T <sub>2</sub>	SPRING-TURN.01 (N/mm <sup>2</sup> )		

Direction of Ratio:	<input type="radio"/>	<input type="radio"/>	Unvariable Requirement	Variable Parameters
Natural Frequency	Static <input type="radio"/>	Dynamic <input type="radio"/>	One Spring Force F1 <input type="checkbox"/> The Associated Length L1 and Spring Rate R1	L1, d, n1
Material	SPRING-MATERIAL.01 G: SPRING-GRA (N/mm <sup>2</sup> ) E: SPRING-ELI (N/mm <sup>2</sup> )		Two Spring Forces F1F2, The Associated Lengths L1L2	L1, d, n1
Surface Treatment	Drawn <input type="radio"/> Polished <input type="radio"/> Phosphated <input type="radio"/> With Coating of Metal <input type="radio"/> Shot-Blasted <input type="radio"/> Intermetallic <input type="radio"/> Irradiated <input type="radio"/> Other <input type="radio"/>		Spring Length without Phos and Spring Rate R1	d, n1
Min. Surface Modification			One Spring Force F1 The Associated Initial Tension	L1
Force of Phosad			One Spring Force F1 Length of Phosad of Spring Spring Length without Phosad L1	d, n1 or n1, d1, d2

Other details about e.g. wire surface modification or tolerance

Hidden/Visible  
 On/Off when it is correctly  
 The Edit/parameters will be changed for acknowledgment of unvariable requirement