

5 - Basic Drawing Methods

Using *LINE*, *CIRCLE*, *ARC*

This lesson uses the very frequently used *LINE*, *CIRCLE* & *ARC* commands to initially explain the basic drawing command techniques and options selection system. The explanations of other drawing commands that follow assume an understanding of all the operational techniques explained here.

Note that generally every command can be operated in three ways -

1. By a drop-down menu
2. By an icon toolbar button
3. By typing a command

In all cases, the command will display prompts for what is required next or options to be selected. These will appear in the Command Bar if it is enabled or else on the status bar. Also, the same options will appear in a pop-up Prompt Box. You can select options either by typing the capitalised letters of the desired option as displayed on the prompt line, or else by clicking the desired option from the Prompt Box menu. The latter is far easier.

In all cases it is important to keep an eye on the command line's prompts because if you enter something when the prompts are asking for something different, you may experience a very baffling result. In the following explanations, the sequence of command prompts is shown.

LINE

Toolbar:  Menu: **Insert, Line** Typed command: **line**, **L** or **I**

The **Line** command can be used to draw a 2D shape comprised of many straight-edged segments, but if segments are created in the same Line command, note that each segment is a separate drawing object.

```
: line  
ENTER to use last point/Follow/<Start of line>:  
Angle/Length/Follow/Undo/<End point>:
```

Press the space bar, enter, or right mouse button to terminate the repeating Line command.

The popup Prompt Box for the options in the above two command steps are shown at right. To specify a point instead of selecting an option just ignore the Prompt Box and define the point either by clicking a point or by typing coordinates for X,Y, e.g. 100,25.85

Close - Instead of pressing Space to terminate the Line command you can press the **C** key or pick Close. It draws an additional final line segment that joins back to the starting point of the current Line command sequence. (Remember the *LINE* command keeps repeating)

Undo - If you make a mistake in defining an end point during the Line command, you can cancel just that one last line segment by typing **U** to use the Line command's **Undo** sub-command. This undoes that last point definition without exiting from the Line command sequence. If you click the Undo button the entire sequence of connected line segments that you just drew will vanish. Repeating commands include an Undo in their options.

Follow - This option will make the new line start at the end of the last-drawn object, and follow on with the same alignment. That is, if the last object drawn was a line, it will follow on along the same direction rather like making the previous line longer. More usefully, if the last object was an arc, it will draw the new line from the end of that arc in a direction tangent to the end of the arc.

Angle & Length – these allow you to type in the length of the line and its angle – extremely useful options. (This is the IntelliCAD 6.4 substitute for AutoCAD's Polar Snap facility which is also in IntelliCAD 6.6.)

ENTER to use last point - When starting a new line or sequence of line segments, instead of responding to the first prompt for a Start Point by defining a point, you can just press Enter (or Space) to make the line start at the end point of the last line that was drawn. This applies in many situations as well as in Line.



Prompt-1



Prompt-2

Note that you can drag the Prompt Box to a new position, and it will appear there next time.

CIRCLE

Menu: **Insert, Circle** Type: **circle** or **c**



Toolbar buttons left to right -

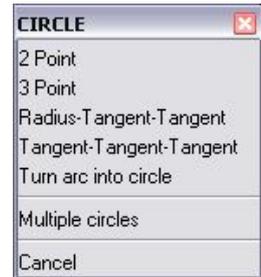
Center-Rad Center-Dia 2 Points 3 Points Radius-Tan-Tan Convert Arc to Circle

Try these examples on a blank screen in AutoCAD as you read the descriptions. We will explain the various command entry methods in some detail here, because the system applies to all commands.

Basic Circle command, or Circle-center-Radius (1st) toolbar button -

```
: circle
2Point/3Point/RadTanTan/TTT/Arc/Multiple/<Center of circle>:
```

Always read the prompts! The sequence shown above, and the following examples, are the prompts that appear if you type CIRCLE, and if you click the Circle Toolbar button. The basic command assumes you want to specify first a center-point, and then a radius. If not, you can pick an option from the list by typing the capitalized letters and Space. That is, you can type: 2P, 3P, RTT, TTT, A or M – or pick them from the popup Prompt Box, which is much easier.



This is the sequence for the 3-Point option -

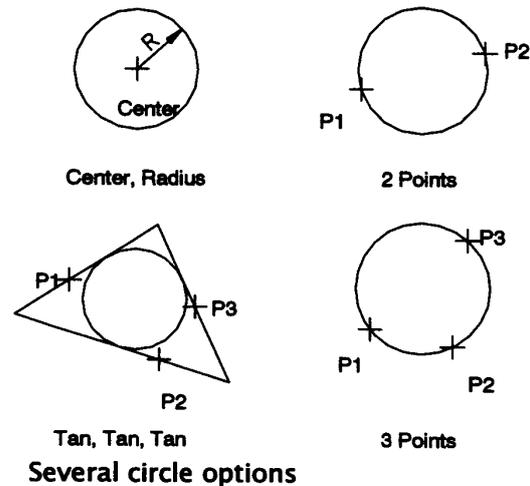
```
: circle
2Point/3Point/RadTanTan/TTT/Arc/Multiple/<Center of circle>:3p (or pick on the menu)
First point on circle: (pick a point)
Second point: (pick a point)
Third point: (pick a point)
```

Note that a 'rubber' circle displays when the cursor is waiting for the third point because that will fix the radius. For the 2P option the rubber circle appears after the first point because the two points of necessity will be the diameter end-points.

The TTT option requires you to pick two points on existing objects to which you want the circle to be tangent, and invokes the Tangent (deferred) object-snap to assist. It then asks for a third point for the circle to pass through, or another tangent point. If the 3rd is not on an object it is used as a pass-through point to set the circle diameter. If the combination proves impossible it will do its best, ignoring one of the points -

```
: circle
2Point/3Point/RadTanTan/TTT/Arc/Multiple/<Center of circle>: ttt
First point on circle:
Second point:
Third point:
```

This is all explained here in some detail because it is the first example of the way the command prompts and options are presented, and the way the menus operate. The same principles apply throughout, so these details will not be explained from here onwards.



ARC



Menu: **Insert, Arc** Type: **arc** or **a**

The above 17 buttons left to right are as follows, reading across & then down to the next line (watch the popup tooltips for each) -

- | | | | | |
|----------------------------|----------------------------|----------------------------|----------------------------|-------------------------|
| 3-points | Length-Start-End | Start-Center-End | Start-End-Center | Tangent Arc |
| Center-Start-Angle | Start-Center-Angle | Start-Angle-Center | Start-End-Angle | Start-Angle-End |
| Center-Start-Length | Start-Center-Length | Start-Direction-End | Start-End-Direction | Start-Radius-End |
| Start-End-Radius | Start-Radius-Angle | | | |

This operates similarly to Circle, but there are more options. Some may seem redundant, but they request features in different sequences, to suit different circumstances. Depending on what existing objects there are and how the arc is to relate to them, all these options are needed to choose the one that suits your needs at that point.

- Tangent** starts tangent to the direction of an existing line-end used as the start point..
- Angle** means the included angle of the arc between the two ends and around its centre point.
- Direction** means the tangent straight-line direction at the end of the arc.
- Length** means the imaginary straight chord line length between the ends of the arc.

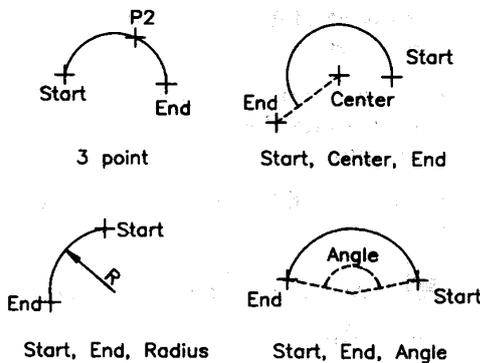
Even with all these options, there are situations when you can't find a suitable method. In those cases it is often easiest to draw a circle instead and then use the Trim or Break command to cut some of it away, leaving the desired Arc. Trim and Break are editing commands, detailed later.

The prompts if you type ARC or pick the Arc 1st toolbar button, are as follows (& Prompt popup at right):

: arc
 ENTER to use last point/Center/Follow/<Start of arc>:



It assumes you want to specify the arc by giving start, mid and end points – or '3-points'. The radius and centre-point will be worked out automatically. All the options seen on the toolbar fly-out can be used from the prompts by picking options as they are presented, but it is not so self-evident. If you want to specify Centre first, type C or pick from the popup.



The diagram at left shows four of the options.

Try drawing arcs by all the methods.

Note that arcs are always drawn counter-clockwise. If you pick points in the wrong direction you don't get what you expect! However, the 3-point method is the exception – it doesn't care because the 3 points control the arc direction.

Some of the prompts are not really obvious enough, but it is best to learn by experiment here. If stumped consult the User Guide or the on-screen Help. Search for Circle.

Four of the many arc options

Undo & Redo



The Undo facility is frequently useful. Clicking the Undo toolbar button (the left-pointing curved arrow button) or typing U at the Command prompt will undo the last command. You can undo right back to the start of the current drawing session.

There is also Redo (the right-pointing curved arrow button), which undoes the last undo.

Note that where a single command draws a sequence of objects, such as Line used to draw a sequence of connected lines, Undo will undo the entire sequence. To undo just the last line-segment, use the U option in the Line command. Some other commands that repeat also have their own undo facility.

Specifying points, lengths and angles

In all the above (and all other draw commands), picking points or ends of lines is always subject to the various constraints that are available in AutoCAD to help you precisely position things. Whenever a prompt asks for a point, you can:

1. Type Absolute Coordinates (eg 25.6,100)
2. Type Relative Cartesian Coordinates (eg @15,10.3)
3. Type Relative Polar Coordinates (eg @150.5<45 - length & angle)
4. Pick randomly on the screen
5. Pick on the screen but constrained by grid-snap
6. Specify length and angle using the Length & Angle options of Line (or the Polar Snap facility in v6.6)
7. Pick a point snapped accurately onto a feature of existing objects using ESNAP.

Options 1 to 5 were detailed in Chapter 4; option 6 is detailed above in this chapter; ESNAPS or Entity-snaps are detailed in Chap 6

Also, specifying lengths & angles by pointing...

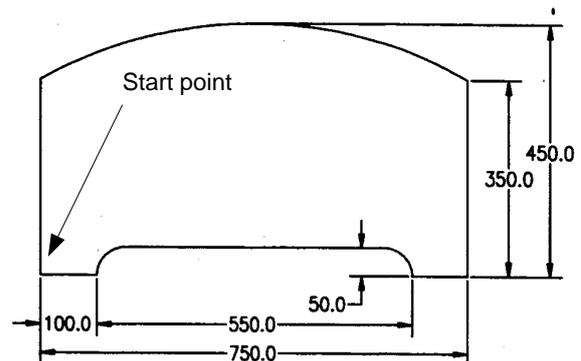
Whenever IntelliCAD prompts for a length, you can, instead of typing a number, pick a point on the screen. It will then prompt for a second point and use the distance between the two picks as the distance value. This can be useful when you are not sure of the accurate value or the prevailing zoom scale of the display and just want something that looks about right. A similar facility exists for specifying angles – pick two points and the angle of the imaginary line between them relative to the zero angle datum is taken as the wanted angle.

Repeating the last command

If you want to use the same command as you just used, press ENTER or SPACE, provided the command line is showing just a colon.

Exercise: Drawing Arcs & Lines

- LINE. 1st point: **0,0**
- 2nd point: **@100<0** (i.e. 100mm at angle 0)
- Use ARC. We want Start-Center-Angle, so -
- Start: **@** (for end of last line - NOT Enter!)
- Pick Center option. Center: **@50,0**
- Pick Angle option. Angle: **-90**
(90 would go anticlockwise)
- LINE. 1st point: **@** (i.e. end of the arc)
- 2nd point: **@450<0** (i.e. 550 minus 2 R50 arcs)
- Continue using similar techniques.



TIPS:

Use Z E (Zoom Extents) whenever your work goes off the screen, or zoom out by the mouse wheel.

You can specify the end of the last drawn element for a start point by @ as above. You can alternatively press *Enter* for the last endpoint as the start of a new arc *provided* you want the arc to start *tangent* to the last line. In this exercise both situations prevail in various places, so use whichever is appropriate.

Use Undo whenever your command gives a wrong result and then figure out a different option or value. If you have to undo and try a step again, you will not be able to use @ for the end of the last object. E-snaps are explained in the next chapter, but you could use that by right-clicking the ESNAP button on the status bar and ticking 'Endpoints'. Then click the end of the arc or line for a start point.

By starting the exercise at 0,0 as described, rather than at a random click, it is possible to fall back on giving absolute coordinates if the @ facility fails because of having used Undo.

The long top arc is a bit tricky. Think about the available options and try. Remember @ is the last point defined. Use the following only if stumped:

Immediately after drawing the 350mm vertical line on the right, use 3-point Arc -

- 1st point: **@** (end of last line - the 350mm vertical one. This only works if that line was just drawn. If you've had to undo a wrong arc, also undo the vertical line and redraw it.)
- 2nd point: **@-375,100** (Half of 750 toward the left & 100 upwards. Note: @ will then be this arc mid point.)
- 3rd point: **@-375,-100** (350 further to left & back down to the same height as the end of the 350 line)

There are 'slicker' ways of drawing this object using tools described later, but this is an exercise on this chapter's topics and the coordinate entry methods in the previous chapter.