Setting up Ahha Software for turning and using tool length offsets

Definitions

Absolute Zero:-

This is the position automatically set by the action of Homing the machine.

It can be set manually but we strongly recommend that the automatic home is used.

It is also the position that relative zero is relative to. By that we mean that if an Absolute Zero is moved by say 10mm then the Relative Zero will be moved by the same amount.

Relative Zero:-

The name given by Fanuc and Ahha to the zero point the programme refers to. If you move X +10. mm in a programme in absolute programme mode you will end up +10mm away from the relative zero. Relative zero is also referred to as programme zero on some CNC controls.

Absolute Display Mode:-

In Absolute display mode the axis position displays show the distance between the current axis position and Absolute Zero. Indicated by Abs being displayed just beneath the axis position values. Pressing the A key toggles between absolute and relative display mode. i.e. every press of the A key switches from one to the other. Absolute display mode is only normally used during setting. At all other times it is much better and sometimes even essential to be in relative display mode.

Relative Display Mode:-

In Relative display mode the axis position displays show the distance between the current axis position and Relative Zero.. Indicated by Rel being displayed just beneath the axis position values. This is the display mode that the control should be in during all normal operations

Setting Up The Machine
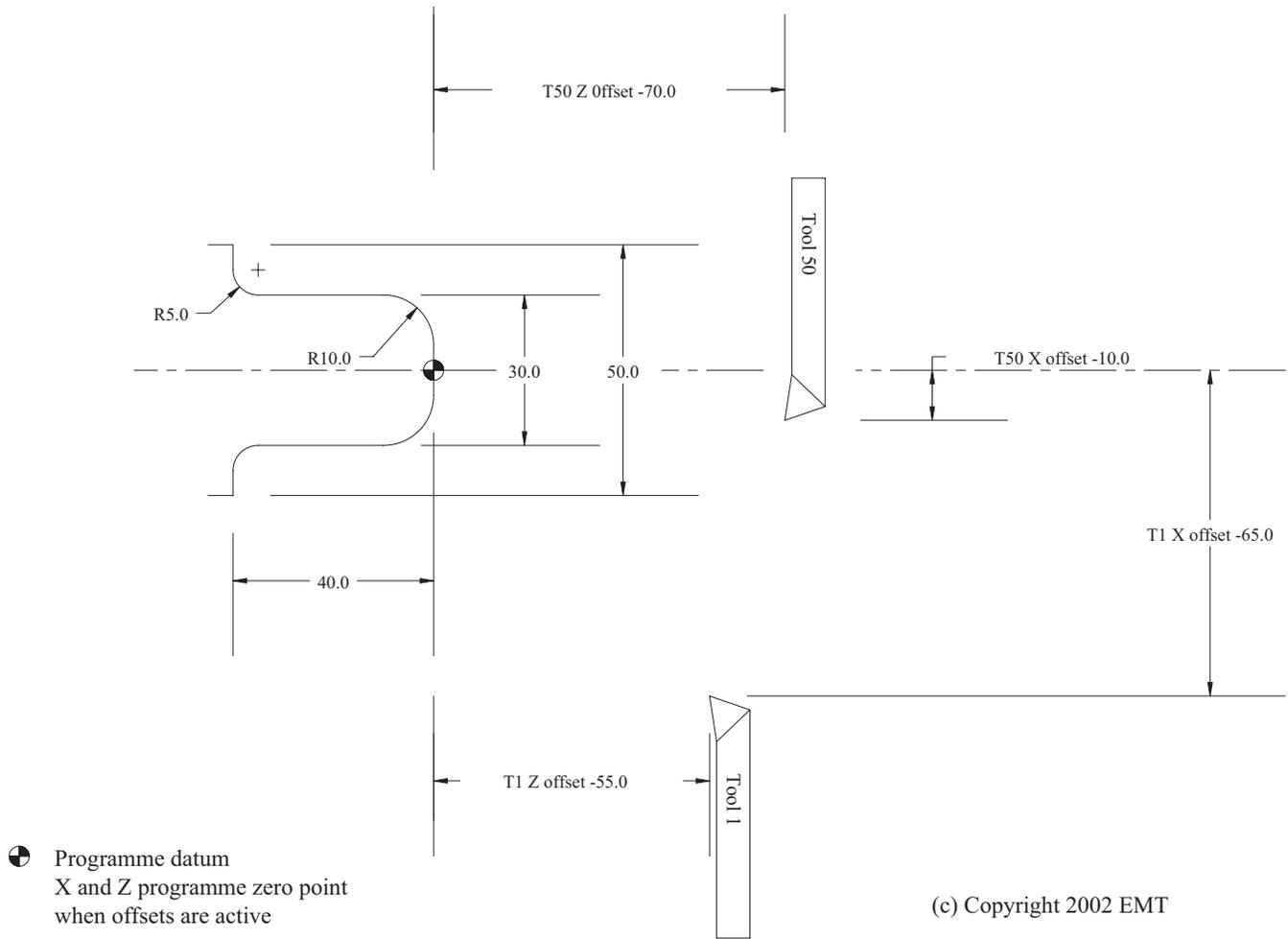
Homing the machine:-

Pressing the Home key and letting the machine set Absolute Zero automatically using the limit switches.

With precision limit switches we have had machines set Absolute Zero repeatedly within .0002".

Absolute Zero can also be set by moving the axis to be set to the required home position and pressing the C key followed by the Enter key followed by the key designating the axis to be set or A if you are setting all axes at the same time.

The most important thing to know before using Ahha tool length offsets when turning is that the Absolute Zero and Relative Zero musts be coincident for turning i.e. both zeros must be in the same place. The ability to move Relative Zero is for milling where it is the only way you can set a Relative Zero to a different place in the machining plane (X and Y) to Absolute Zero. In turning you have to use the tool length offsets because of the different position of each tools cutting edges when the slides are at Absolute Zero. You can check that the machine is set correctly by pressing the G key. The machine will move to the Relative Zero. Switch between absolute display and relative display by pressing the A key. The axis displays should all read 00.0000. If they do not both read 00.0000 home the machine again and while you are still at the home position press the S key followed by the Enter key followed by the A key.

Drawing #1 Ahha Tool Length Compensation Using Absolute Reference Set With Home Key

T50 Z 0ffset -70.0

Tool 50

R5.0

+

R10.0

30.0

50.0

T50 X offset -10.0

40.0

T1 X offset -65.0

T1 Z offset -55.0

Tool 1

Programme datum
X and Z programme zero point
when offsets are active

(c) Copyright 2002 EMT

Note:-

This applies to a front tool post machine. X plus moves to the front of the machine towards the operator.

In our set up we use consistent tool numbers

All tools numbered 1 to 49 go in front tool post

All tools numbered 50 and above go in rear tool post

Tools are shown drawn at Absolute Zero position which on a lathe is also be the Relative Zero position with slides to the front and right (X + Maximum Z + Maximum)

Drawing #2 Ahha Tool Length Compensation Using Programme Reference Set Manually

Tool 50

T50 Z 0ffset -70.0

T50 X offset 55.0

R5.0

+

R10.0

30.0

50.0

T1 X offset 0.0

Tool 1

40.0

T1 Z offset -55.0

Programme datum
X and Z programme zero point
when offsets are active

Note:-

This applies to a front tool post machine. X plus moves to the front of the machine towards the operator.
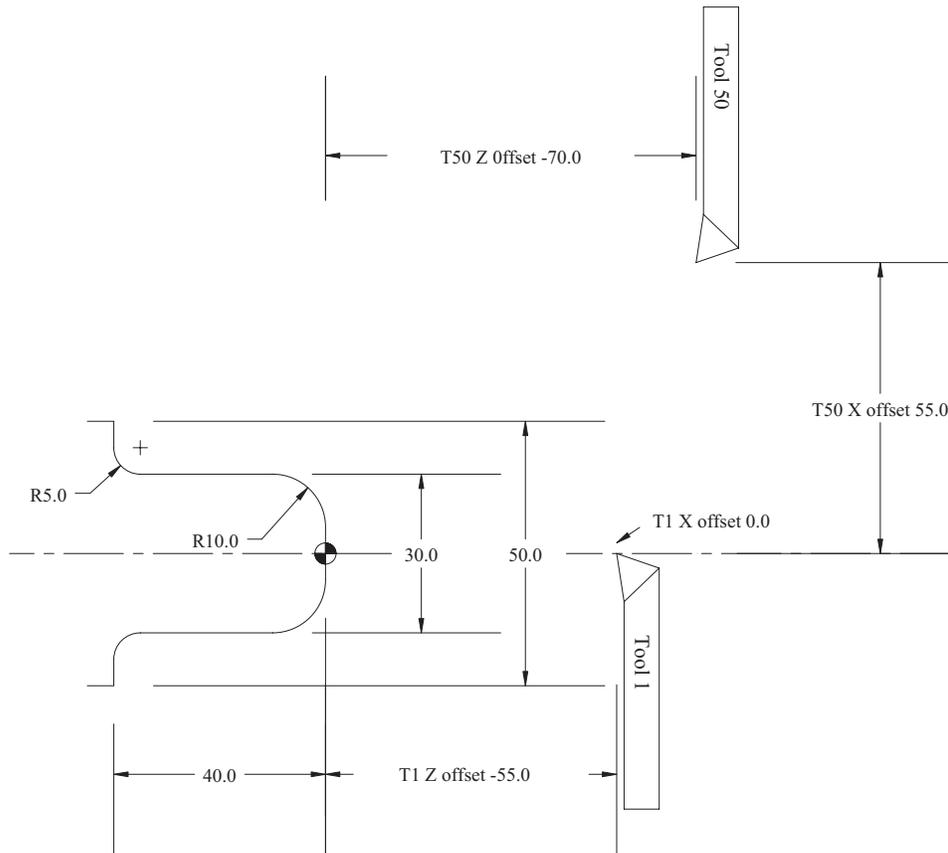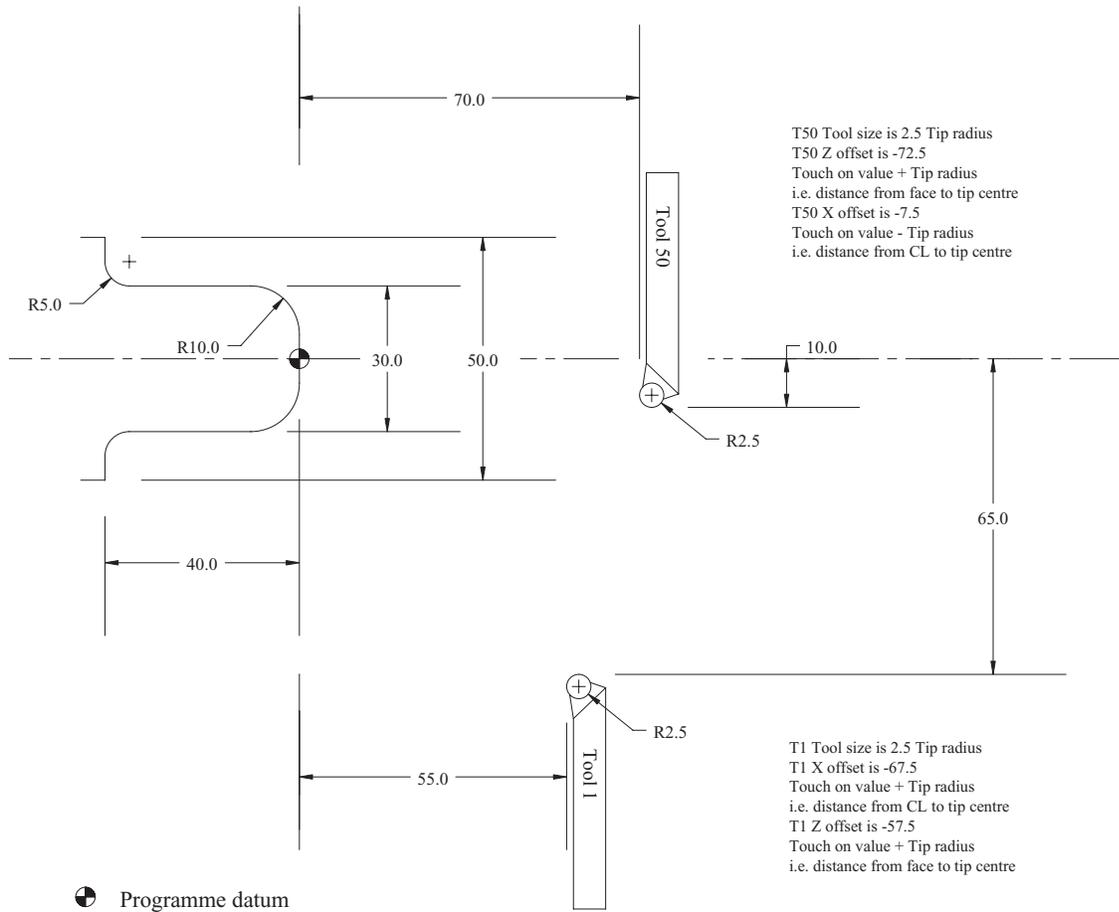
In our set up we use consistent tool numbers

All tools numbered 1 to 49 go in front tool post

All tools numbered 50 and above go in rear tool post

Tools shown drawn at a manually set Absolute Zero position which on a lathe is also be the Relative Zero position with slides to Right (Z + Maximum) but the cross slide positioned with longest front tool positioned at the centre of the workpiece.

Drawing #3 Using Ahha Tool Tip Radius Compensation

70.0

T50 Tool size is 2.5 Tip radius
T50 Z offset is -72.5
Touch on value + Tip radius
i.e. distance from face to tip centre
T50 X offset is -7.5
Touch on value - Tip radius
i.e. distance from CL to tip centre

Tool 50

R5.0

+

R10.0

30.0   50.0

10.0

R2.5

65.0

40.0

R2.5

Tool 1

55.0

T1 Tool size is 2.5 Tip radius
T1 X offset is -67.5
Touch on value + Tip radius
i.e. distance from CL to tip centre
T1 Z offset is -57.5
Touch on value + Tip radius
i.e. distance from face to tip centre

Programme datum
X and Z programme zero point
when offsets are active

(c) Copyright 2001 EMT

```
; Drawings 1 & 2 Front tool T1
; programme details
;
;
;
;
 G21 *                                ; Metric programme
 G90 *                                ; In absolute
 G40 *                                ; Make sure radius offsets are off
 T0000 *                              ; use TO offset as tool change position
 G0 Z0.0 X0.0 *                       ; Tool change position
 T0101 *                              ; Set current tool offset
 M06 *                                ; Prompt for tool change
 F25. *                               ; Set feed rate for current tool
 S1250. *                             ; Set spindle speed
 M03 *                                ; Turn spindle on
 G00 X0.0 Z0.0 *
 G01 X5.0 *
 G03 X15.0 Z-10.0 I0.0 K-10.0 *
 G01 Z-35.0 *
 G02 X20.0 Z-40.0 I5.0 K0.0 *
 G01 X25.0 *
 G00 X30.0 Z5.0 *
 T0000 *                 ; Use TO offsets as tool change position
 G00 Z0.0 X0.0 *         ; Park position at end of programme
 M05 *                   ; Turn spindle off
 S0. *                   ; Set S to 0
 M30 *                   ; End of programme, rewind, prompt to run again


; Description here for file window
; programme details
;
;
;
;
 G21 *                                ; Metric programme
 G90 *                                ; In absolute
 G40 *                                ; Make sure radius offsets are off
 T0000 *                              ; use TO offset as tool change position
 G0 Z0.0 X0.0 *                       ; Tool change position
 T5050 *                              ; Set current tool offset
 M06 *                                ; Prompt for tool change
 F25. *                               ; Set feed rate for current tool
 S1250. *                             ; Set spindle speed
 M03 *                                ; Turn spindle on
 G00 G43 X0.0 Z0.0 *
 G40 G01 X-5.0 *
 G02 X-15.0 Z-10.0 I0.0 K-10.0 *
 G01 X-15.0 Z-35.0 *
 G03 X-20.0 Z-40.0 I-5.0 K0.0 *
 G01 X-25.0 *
 G00 X-30.0 Z5.0 *
 T0000 *                 ; Use TO offsets as tool change position
 G00 Z0.0 X0.0 *         ; Park position at end of programme
 M05 *                   ; Turn spindle off
 S0. *                   ; Set S to 0
 M30 *                   ; End of programme, rewind, prompt to run again
```

```
; Drawing 3 Rear tool T50
; programme details
;
;
;
 G21 *                             ; Metric programme
 G90 *                             ; In absolute
 G40 *                             ; Make sure radius offsets are off
 T0000 *                           ; use TO offset as tool change position
 G0 Z0.0 X0.0 *                    ; Tool change position
 T5050 *                           ; Set current tool offset
 M06 *                             ; Prompt for tool change
 F25. *                            ; Set feed rate for current tool
 S1250. *                          ; Set spindle speed
 M03 *                             ; Turn spindle on
 G00 G41 X0.0 Z0.0 *
 G01 X-5.0 *
 G02 X-15.0 Z-10.0 I0.0 K-10.0 *
 G41 G01 X-15.0 Z-35.0 *
 G03 X-20.0 Z-40.0 I-5.0 K0.0 *
 G41 G01 X-25.0 *
 G00 G40 X-30.0 Z5.0 *
 T0000 *              ; Use TO offsets as tool change position
 G00 Z0.0 X0.0 *      ; Park position at end of programme
 M05 *                ; Turn spindle off
 S0. *                ; Set S to 0
 M30 *                ; End of programme, rewind, prompt to run again
```