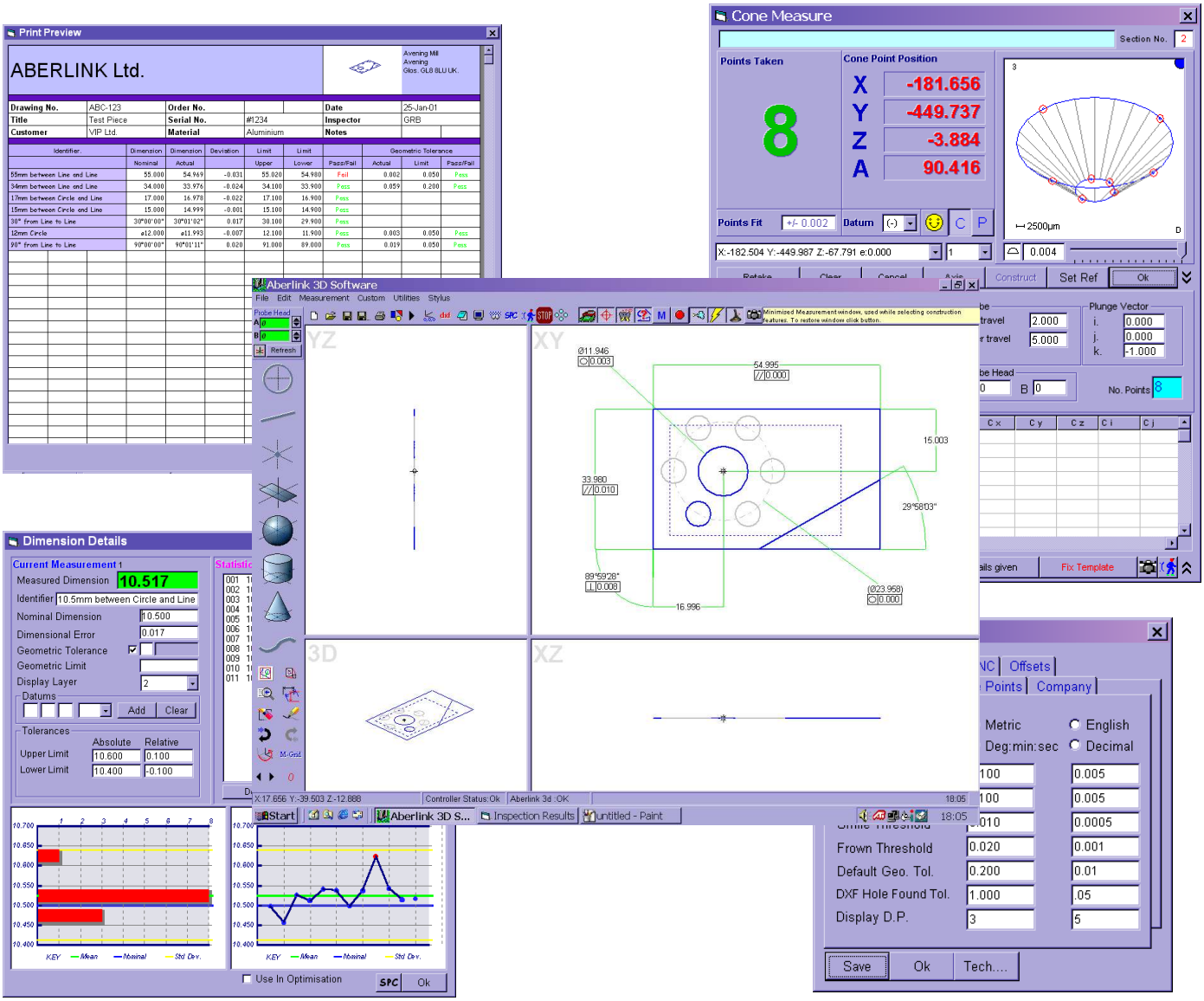
Mk3 MEASUREMENT SOFTWARE MANUAL CNC



ABERLINK LTD. VATCH LANE EASTCOMBE GLOS. GL6 7DY UK

www.EngineeringBooksPdf.com

INDEX

1 .0 INTRODUCTION TO CMM MEASUREMENT 13

1.1 Preparation & general hints for solving measurement problems 13

2.0 STARTING THE SOFTWARE 14

2.1 Before starting the software 14

3.0 EXPLORING THE MAIN SCREEN 17

4.0 THE JOYSTICK 23

4.1 Movement X, Y and Z 24

4.2 Taking Points (at constant feed) 24

4.3 Indexing the Probe 24

4.4 Locking the Axes 25

4.5 The Speed Control 25

4.6 The Trigger Button 25

5.0 DATUMING THE PROBE 26

5.1 The Probe Datumming Button 26

5.2 Datummıng the Probe 27

5.3 Using a non-indexible probe 32

5.4 Manually datum star stylus 34

5.4 Automatically datum star stylus 34

5.5 Saving Probe list 36

5.6 Datumming using a Disc Stylus 37

5.7 Tool changing rack 37

6.0 ALIGNING THE AXES OF A COMPONENT 40

6.0.1 Introduction 40

www.EngineeringBooksPdf.com

6.0.2 Example of traditional part alignment using a surface table, straight edge & pins. 40
6.0.3 Example of alignment using Aberlink 3D 41

6.1 Basic Alignment 49

6.2 Plane & two circles Alignment 51

6.3 Aligning components using a cylinder 52

6.4 Drive shaft (aligning along shaft's centre line) 54

6.5 Optimising Alignment 56
6.5.1 Optimising to Horizontal and Vertical Dimensions 56

6.6 Optimising to a DXF File 58

6.7 3,2,1 Optimised Alignment 59

Y 59

Z 59

7.0 CREATING A CNC PROGRAM – A GENERAL INTRODUCTION 63

7.1 Teach and Repeat 63

7.2 Off-Line Programming 63

8.0 THE PLANE UNIT 64

8.1 Measuring Planes Directly 64

8.2 Teach and Repeat 64

8.2. Off-Line Programming 67

8. 3 The 'Template from Measured Data calculator' button 72

8.4 When to OK a plane, and when to reference it 73

8.5 Constructing a Plane through previously measured points 75

8.6 Constructing a Plane offset from another Plane 77

8.7 Constructing a mid plane between two measured planes 78

9.0 THE LINE UNIT 80

9.1 Measuring Lines Directly 80
9.1.1 When to OK a line, and when to reference it. 84
9.1.2 Measuring a line in a user defined plane 86

www.EngineeringBooksPdf.com

9.2 Constructing Lines 88
9.2.1 Constructing Lines through 2 or more points 88
9.2.2 Constructing a Line at the Intersection of 2 Planes 89
9.2.3 Constructing a Centre Line between 2 Other Lines 91
9.2.4 Constructing a Tangent Line between 2 circles or a point & circle 92
9.2.5 Constructing a Gauge Line of Fixed Length between 2 other Lines 95

10.0 THE SPHERE UNIT 98

10.1 Displaying the centre position in polar co-ordinates 100

10.2 When to OK a Sphere, and when to reference it. 101

11.0 THE CIRCLE UNIT 103

11.1 Measuring Circles Directly 103
11.1.1 Displaying the centre position in polar co-ordinates .. 108
11.1.2 Off-line Programming. 108
11.1. 3 The 'Template from Measured Data' button 113
11.1.4 When to OK a Circle, and when to reference. 114
11.1.5 Measuring a circle in a user defined plane 115

11.2 Constructing Circles 116
11.2.1 Constructing a Pitch Circle Diameter 116
11.2.2 Constructing a Circle at the Intersection of Two Features 118
11.2.3 Constructing of gauge circle of a fixed diameter between two points or lines 120

12.0 THE POINT UNIT 122

12.1 Measuring Points Directly 122

12.1.1 Displaying the centre position in polar co-ordinates 126
12.1.2 Off Line Programming 126
12.1.3 Measuring with Compensated Approach 129
12.1.4 When to OK a Point, and when to reference it 131

12.2 Constructing Points 131
12.2.1 Entering the Co-ordinates for the Point 131
12.2.2 Constructing points between features with a single intersection 133
12.2.3 Constructing Points between features when there is more than one intersection 136
12.2.4 Constructing Points between two cylinders 137
12.2.5 Constructing a Point at the Mid-Point or the end of a Line 141

www.EngineeringBooksPdf.com

13.0 THE CYLINDER UNIT 145

13.1.2 Off-Line Programming 148

13.1 3 The 'Template from Measured Data' button 153

13.1.4 When to OK a Cylinder, and when to reference it. 154

14.0 THE CONE UNIT 158

14.1 Measuring cones without defining their axis 158

14.2 Measuring cones having defined their axis 162

14.3 Off Line Programming 163

14.4 The 'Template from Measured Data' button 167

14.5 When to OK a Cone, and when to reference it. 168

14.6 Measuring the maximum diameter of a cone. 170

14.7 Determining the Position on a cone of a given diameter. 171

15.0 THE CURVE MEASURE UNIT 172

15.1 Creating a Curve Measure Program Unit 172

15.2 Measuring a 2D curves in a fixed plane (XY,ZX,YZ) using the curve button. 172

15.3 Compare a Curve Profile to nominal DXF Profile 177

15.4 Measuring a 3D surface area using the curve button 183

15.5 Measuring multiple points using the curve button 189

16.0 ERASING DATA POINT 191

16.1 Retake 191

16.2 Clear 192

16.3 Delete 193

17.0 MANIPULATING THE GRAPHICS 194

17.1 Zoom Out 194

17.2 Zoom Full 194

17.3 Zoom In 194

www.EngineeringBooksPdf.com

17.4 Re-sizing & re-positioning the Graphics windows 196

17.5 Erase 197

17.6Undo and Redo 197

17.7 Redraw 198

18.0 SHIFTING, ROTATING AND CANCELLING REFERENCES 199

18.1 Applying Rotations or Shifts 199

18.2 Machine Grid (Cancelling References) 201

18.3 Optimising Alignment 201
18.3.1 Nominal optimization:- 202
18.3.2 Tolerance optimization:- 203

19.0 BRINGING UP DIMENSIONS ON THE SCREEN 204

19.1 Selecting Features 205

19.2 Using the Feature Select Buttons 206

19.3 Aligned Dimensions 208
19.3.1 Three Dimensional 208
19.3.2 Two Dimensional 211

19.4 Horizontal and Vertical Dimensions 213

19.5 Angular Dimensions 214
19.5.1 Three Dimensional 214
19.5.2 Two Dimensional 215

19.6 Diameter Dimensions 216

19.7 Dimensioning between two Lines or Planes 217

19.8 Dimensioning to a Cone 218

19.9 Aligned to Feature & Ant-aligned to Feature 221

19.10 Leader option from ‘Dimension type drop down menu’ 222

19.11 Circle Max & Min Measure Option 222

19.12 Setting Nominal Values and Tolerances for Dimensions 223

20.0 GEOMETRICAL TOLERANCES 227

Geometric Characteristic Symbols 231

www.EngineeringBooksPdf.com

20.1 Straightness....231
20.2 Flatness....231
20.3 Roundness or Circularity....232
20.4 Cylindricity....232
20.5 Parallelism....233
20.6 Angularity....233
20.7 Perpendicular (Squareness)....233
20.7 Concentricity....234
20.8 Total Runout....234
20.8.1 Run out....236
1. Plane to Cylinder....236
2. Circle to Cylinder....236
20.9 True Position....237
20.9.1 True Position relative to specified datum(s)....238
20.9.2 Maximum Material Condition....240
20.10 Symmetry....242
21.0 GENERAL INFORMATION ABOUT MEASURED UNITS....244
21.1 Recalling a measured Unit....244
21.2 Labelling Specific Measurement Units....244
21.3 Counting Down....245
22.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS....247
23.0 SAVING MEASUREMENTS....250
23.1 Save As....250
23.2 Save File....251
24.0 STARTING A NEW INSPECTION....252
25.0 OPENING A PREVIOSLY SAVED INSPECTION....254
26.0 PRINTING RESULTS....255

www.EngineeringBooksPdf.com

26.1 Graphic Details 256
26.2 Tabulated Units 257
26.3 Tabulated Dimensions 258
26.4 Point Positions 260
26.5 Feature Profile 260
26.6 Multiple Components 261

27.0 THE PLAY FUNCTION – RUNNING A PROGRAM AUTOMATICALLY 263
27.1 The Safe Volume 263
27.2 The Approaches 264
27.3 The Workpiece Co-ordinate (or WPC) 266
27.4 The CNC Program Window 267
27.4.1 The Program Tree 267
27.4.8 Run Program and Cancel (8) 271
27.5 The work piece co-ordinate (WPC or Work off set) 273
27.6 Setting rotations with WPC 274
27.6.1 Setting Multiple Workpiece Co-ordinates (multiple components) 275
27.6.2 Automatically setting work offsets for a grid of components 275
27.6.3 The Safe Volume Values 276

28.0 SOFTWARE SET UP 278
28.1 Units Tab 279
28.1.1 Metric or English 279
28.1.2 Display Angles as Deg, Min, Sec or as a decimal 279
28.1.3 Round Nominals To 279
28.1.4 Default Tolerance 279
28.1.5 Smile / Frown Threshold 280
28.1.6 DXF Hole Found Tolerance 280
28.1.7 Default True Pos. Tolerance 280
28.1.8 Display D.P. 280
28.1.10 The ISO 2768 Tolerance standard 280
28.2 Display Tab 282
28.2.1 Selection Box Size 282
28.2.2 Point Size (when drawing) 282
28.2.3 Display Font 282

www.EngineeringBooksPdf.com

28.2.4 Max Font Size 283
28.2.5 Display Averaging 283
28.2.6 Polar Co-ordinates 283

28.3 Feature Points 284

28.3 Feature Points 284

28.4 Company 285

28.5 General 286
28.5.1 Temperature (deg C) 286
28.5.2 Material 287
28.5.3 Thermal Expansion coefficient (PPM) 287
28.5.4 Results Folder 287
28.5.5 Probe Override 287
28.5.6 Auto Point Capturetime 288

28.6 Ref Ball Tab 288
28.6.1 Reference Ball 288
28.6.2 Ref. Dia 289
28.6.3 Threshold 289
28.6.4 No. of points 289
28.6.5 Stylus Len. and Stylus Dia. 289
28.6.6 Stem Vector and Stem Dia. 289
28.6.7 Stand Off 289
28.6.8 B Angle Set 289
28.6.9 How to re-calibrate the reference ball position. 290

28.7 CNC Tab 291
28.7.1 Default Approach Distance 291
28.7.2 Message Box Count down Time 291
28.7.3 Default Over Travel 291
27.7.4Message Box Count down Time 292

28.8 Offsets Tab 292

29.0 SPC BATCH INFORMATION 294

30.0 THE JOG MACHINE BUTTON 298

31.0 THE STOP BUTTON 299

32.0 THE PARK FUNCTION 300

33.0 DISPLAYING THE PROGRAM TEMPLATE 301

34.0 THE AIR SWITCH 303

www.EngineeringBooksPdf.com

35.0 THE MOTOR DRIVES BUTTON 304

36.0 THE CCD CAMERA SYSTEM BUTTON 305

37.0 THE RING / GRID ARRAY 306

37.1 A Ring Array 306

37.2 A Grid Array 308

38.0 DISPLAYING PROBE CONTACT POINTS 311

38.1 Moving the measurement points 311

38.2 Adding 'Move Via' graphically 313

39.0 INSERTING A 'MOVE VIA' INTO A MEASURING ROUTINE 315
39.1 The Joystick Method 316
39.2 On the Screen Method 316
39.3 Edit Measurement Grid Method 317

40.0 THE STORE POINTS FUNCTION 320

41.0 THE FEATURE PREDICT FUNCTION 321

42.0 CHANGING THE COLOUR OF A FEATURE 322

43.0 INPUTTING AND OUTPUTTING DXF FILES 324

43.1 Import Hole Positions 325

43.2 Import Program Template 325

43.3 Import As Measured 325

43.4 Import As Curve Profile 325

44.0 THE PLAY FUNCTION 332

44.0.1 Useful Hints 336

44.1 More information on the Program Tree 337

44.2 Collecting SPC Batch Information 337

45.0 INSPECTION NOTES 339

45.1 Adding Text 339

www.EngineeringBooksPdf.com

45.2 Adding Images.....340

46.0 DISPLAY LAYERS.....341

47.0 CHANGE HISTORY 344

www.EngineeringBooksPdf.com

www.EngineeringBooksPdf.com

1 .0 Introduction to CMM measurement

1.1 Preparation & general hints for solving measurement problems

Before you start measuring with your CMM you must go through some preparation functions. These functions include:

- Give some thought to how you are going to position and clamp the part to be measured.
  Reason: *- Ensure good accessibility, you can access all the features you want measure.*

- Decide which stylus & extension you are going to need and if you are using an indexible head which probe angles you are going to need to measure the part before you start.
  Reason: *- Save time while programming.*
  -Saves you having to move the part.

- It is good working practice to write the feature numbers on the drawing.
  Reason: *- Easy verification of the inspection report.*
  -Easy guidance when running a part inspection program.

- It is good working practice to assign the datum letters to measured features as they are shown in the drawing.
  Reason: *- Easy verification of the inspection report.*
  -Easy true positions, just enter the datum as shown on drawing.

www.EngineeringBooksPdf.com

2.0 Starting the software

2.1 Before starting the software

- Check the air supply
  Before starting check your CMM has a minimum clean dry air supply of 3.9 bar (55 psi) with an airflow greater than 20l/min.
  Check the filters (pots) are free of oil and water contamination (***oil & water must never get through to air bearings as this could damage your machine***)

- **Check the computer is switched on and is displaying windows correctly without showing any error messages.**

- **Check the slide ways of the CMM for any damage or contamination.**

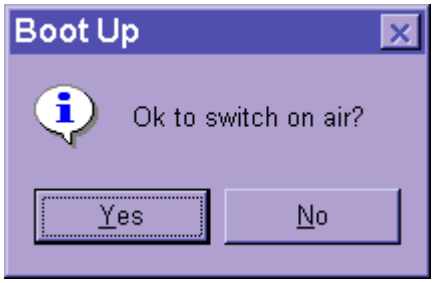
To start the software,
Select 'Start' / 'Programs' / 'Aberlink3D' from the Windows XP Desktop.

Fig. 1.0-1 Showing the Windows XP Start Path to the Aberlink 3D software

Or an alternative way to start the software is to double click on the Aberlink 3D icon on the windows desktop.

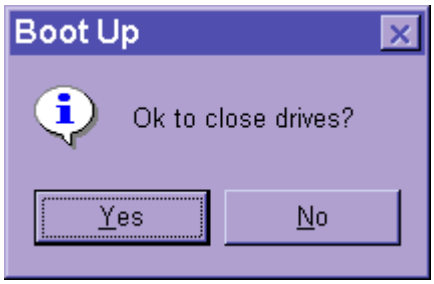
www.EngineeringBooksPdf.com

When the Aberlink 3D software starts. The first message that will appear will be:



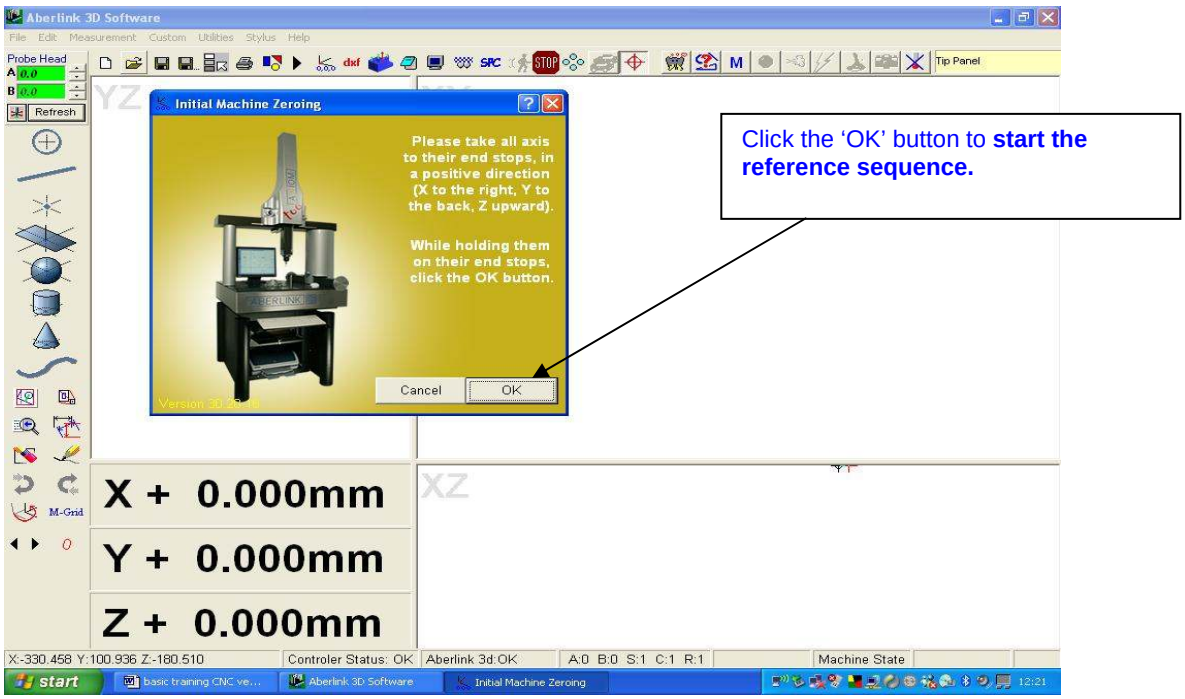
Click 'OK' to switch the air on automatically. at this stage in the start up sequence it is possible to move the machine by hand in to a safe area.

After you click 'OK' another message will then be displayed on the screen:



Again click 'OK to engage the motor drives with the motors engage. the machine will now move with the joystick.

The following window will then be displayed on the screen:



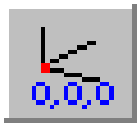
www.EngineeringBooksPdf.com

Each time that the machine is switched on, before you can start to take measurements, the machine has to be referenced. This is done because each of the scales on the machine needs knows where it is with respect to the global error map.

To reference the machine, **click 'OK'**. The machine will now move in a positive direction (Z first, upwards, followed by X and then Y) to find each axis reference mark. When the machine is successfully referenced the above reference window will disappear.

Note:- that before clicking on 'OK' you should make sure that the machine is on the negative side of each of the 3 reference marks.

Should you ever need to re-reference the machine, this can be done at any time afterwards by clicking on the 'Reference Machine' button from the top tool bar on the main screen:



This will bring back up the Machine Reference Screen as previously described.

www.EngineeringBooksPdf.com

3.0 EXPLORING THE MAIN SCREEN

The Aberlink 3D software has been written to be extremely intuitive, based around a graphical interface, operating under Windows XP.

As a component is inspected, a graphical representation of it is built up in the three 2 dimensional views in the main screen. All the functionality of the software is available from the main screen, and this will only ever bring up one further layer, or window. In this way the screen can never become too confusing, as closing the top window will always return to the main screen.

Dimensions can be ‘picked off’ the screen in a way that corresponds directly to the way the component has been dimensioned on the drawing.

It is therefore worthwhile to take a few minutes to become familiar with the buttons available in the main screen.

www.EngineeringBooksPdf.com

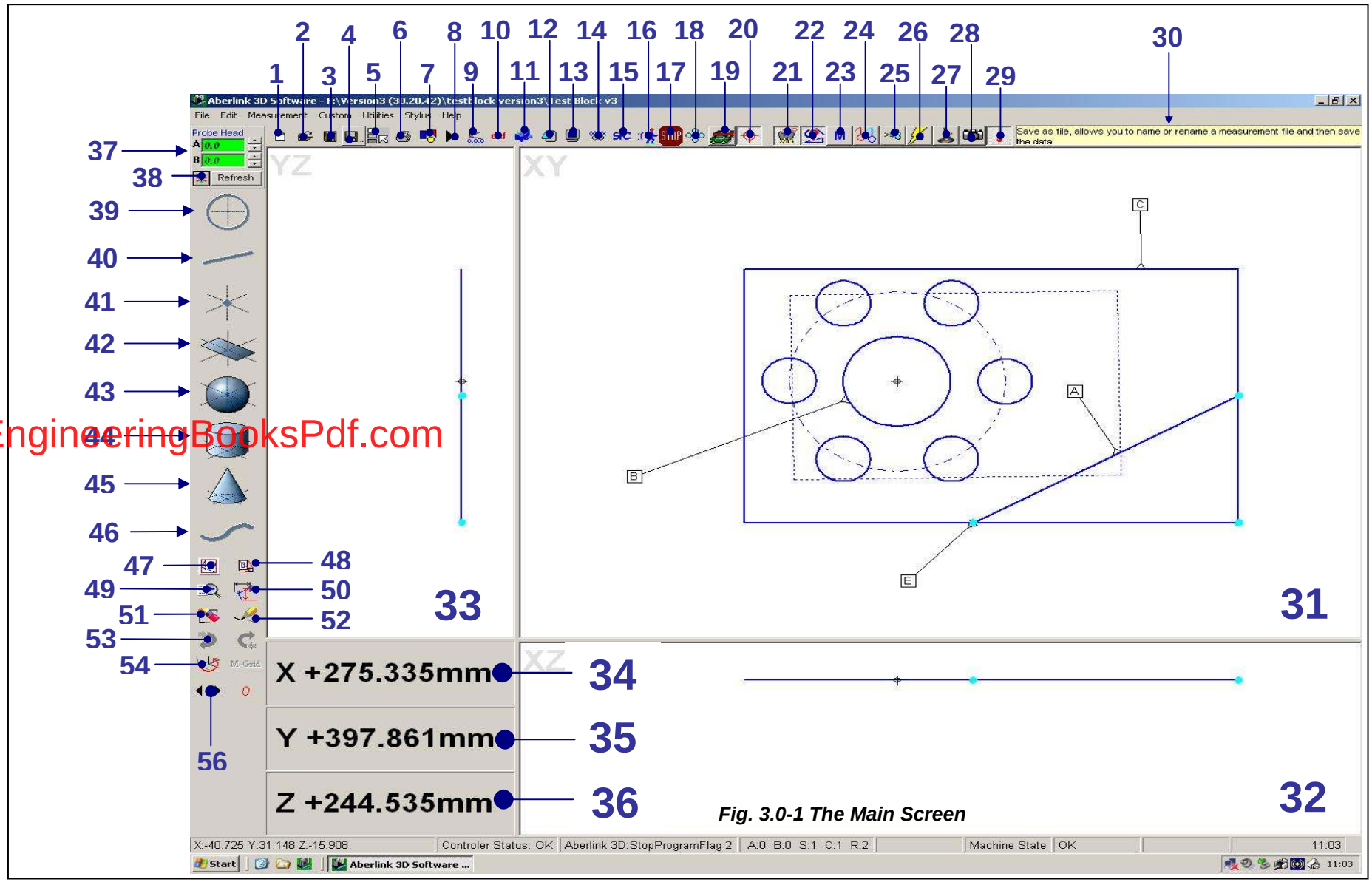


Fig. 3.0-1 The Main Screen

www.EngineeringBooksPdf.com

1. New Inspection (24.0 STARTING A NEW INSPECTION)
Click on this button when you wish to start a new inspection.

2. Open File (25.0 OPENING A PREVIOSLY SAVED INSPECTION)
Click on this button when you wish to recall a previously saved inspection.

3. Save File (23.0 SAVING MEASUREMENTS)
This button will save an inspection under its existing name.

4. Save As (23.1 Save As)
Use this button when you wish to save an inspection giving it a new name.

5. Name & save File ( )
Use this button when you wish to save an inspection with password protection.

6. Print (26.0 PRINTING RESULTS)
Click on this button when you wish to produce a hard copy inspection report.

7. Set Up (28.0 SOFTWARE SET UP)
For setting up the user definable parameters that the software uses.

8. Play (44.0 THE PLAY FUNCTION)
Prompts the measurement sequence for multiple components.

9. Reference Machine (2.0 Starting the software)
This button allows the user to re-reference the machine at any time.

10. DXF File & Curve File save / open (43.4 Import As Curve Profile)
This button will save or open data / curves in DXF format for use with CAD or curve compare function.

11. Import and measure against solid CAD models
This button allows you to import a solid CAD model and measure against it **(this is an optional software module).**

12. Inspection Notes (45.0 INSPECTION NOTES)
This allows text and images to be stored with inspections.

13. Display Drawing Layers (46.0 DISPLAY LAYERS)
This button turn layers on and off the 'Display Drawing Layers'

14. Work piece co-ordinate (27.3 The Workpiece Co-ordinate (or WPC))
This button opens the work piece co-ordinate (WPC) window

www.EngineeringBooksPdf.com

15. External SPC Data Link (chapter)
This button allows you export data to third party SPC software
(this is an optional software module).

16. Jog Button (30.0 THE JOG MACHINE BUTTON)
This button allows you move the CMM without using the joystick.

17. Stop Button (31.0 THE STOP BUTTON)
This button stops a CNC program.

18. Ring array Button (37.0 THE RING / GRID ARRAY)
This button opens the ring array window.

19. Park CMM Button (32.0 THE PARK FUNCTION)
This button Parks the CMM.

20. Geometric Tolerance (20.0 GEOMETRICAL TOLERANCES)
This button will display relevant geometric tolerances on the screen.

21. Store Points (40.0 THE STORE POINTS FUNCTION)
This function will store measurement points and then apply them to a feature when called up retrospectively.

22. Feature Predict (41.0 THE FEATURE PREDICT FUNCTION)
This function will predict what feature to fit to taken measurement points.

23. Display measured Details or template Details (33.0 DISPLAYING THE PROGRAM TEMPLATE)
This button switches the display between template and measured details.

24. Display probe contact points (38.0 Displaying probe contact points
This button displays the probe path & points.

25. Air on (34.0 THE AIR SWITCH)
This button turns the air on or off.

26. Drives on or off(35.0 THE MOTOR DRIVES BUTTON)
This button turns the dives on or off.

27. Joystick (4.0 THE JOYSTICK)
This button turns the joystick on or off.

28. Camera on/off Button
This button is will activate the camera and allow you to measure the work piece visually. **(this is an optional software module)**

29. Touch Probe on / off Button
This button will activate the touch probe and allow you to measure the work piece using point data.

www.EngineeringBooksPdf.com

30. Tool Tip Bar
Gives useful hints when performing functions.

31. XY Graphics Area (17.0 MANIPULATING THE GRAPHICS)
The XY view of the measured features will appear in this area.

32. XZ Graphics Area (17.0 MANIPULATING THE GRAPHICS)
The XZ view of the measured features will appear in this area.

33. YZ Graphics Area (17.0 MANIPULATING THE GRAPHICS)
The YZ view of the measured features will appear in this area.

34. X Axis Readout (22.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS)
Can be used for performing quick check measurements.

35. Y Axis Readout (22.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS)
Can be used for performing quick check measurements.

36. Z Axis Readout (22.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS)
Can be used for performing quick check measurements.

37. Probe Datum Button(Chapter)
This is used to datum new styli and probe positions.

38. Probe Head Icon (Chapter)
This will change the Probe Datum Button between probe head A & B angles and the star stylus .

39. Circle Measure Button (11.0 THE CIRCLE UNIT)
Used when measuring or constructing arcs, holes or circular bosses.

40. Line Measure Button (9.0 THE LINE UNIT)
Used when measuring or constructing straight lines.

41. Point Measure Button (12.0 THE POINT UNIT)
Used when measuring or constructing single points.

42. Plane Measure Button (8.0 THE PLANE UNIT)
Used when measuring or constructing flat faces.

43. Sphere Measure Button (10.0 THE SPHERE UNIT)
Used when measuring internal or external spheres.

44. Cylinder Measure Button (13.0 THE CYLINDER UNIT)
Used when measuring holes or circular bosses in 3 dimensions.

www.EngineeringBooksPdf.com

45. Cone Measure Button (14.0 THE CONE UNIT)
Used when measuring conical shapes in 3 dimensions.

46. Curve measure Button (15.0 THE CURVE MEASURE UNIT)
Used when measuring curves & profiles

47. Zoom Full (17.2 Zoom Full)
Will zoom the graphics areas to cover the full size of the table.

48. Zoom Out (17.1 Zoom Out)
Will zoom the graphics areas out by a fixed percentage.

49. Zoom Last (17.3 Zoom In)
Will return the graphics areas to their last view.

50. Dimension Type (Chapter)
Can be used to define the type of dimension brought up onto the screen

51. Erase (17.5 Erase)
Can be used to remove any given feature or dimension.

52. Auto Zoom Redraw (17.7 Redraw)
Will automatically zoom the graphics areas to fit the existing features and dimensions.

53. Undo (17.6Undo and Redo)
Will undo the last action performed.
Redo
Will redo the last action undone.

54. Grid Shift (18.0 SHIFTING, ROTATING AND CANCELLING REFERENCES)
Allows a rotation or translation of the measured features, or an optimum alignment to minimise the errors for selected dimensions.

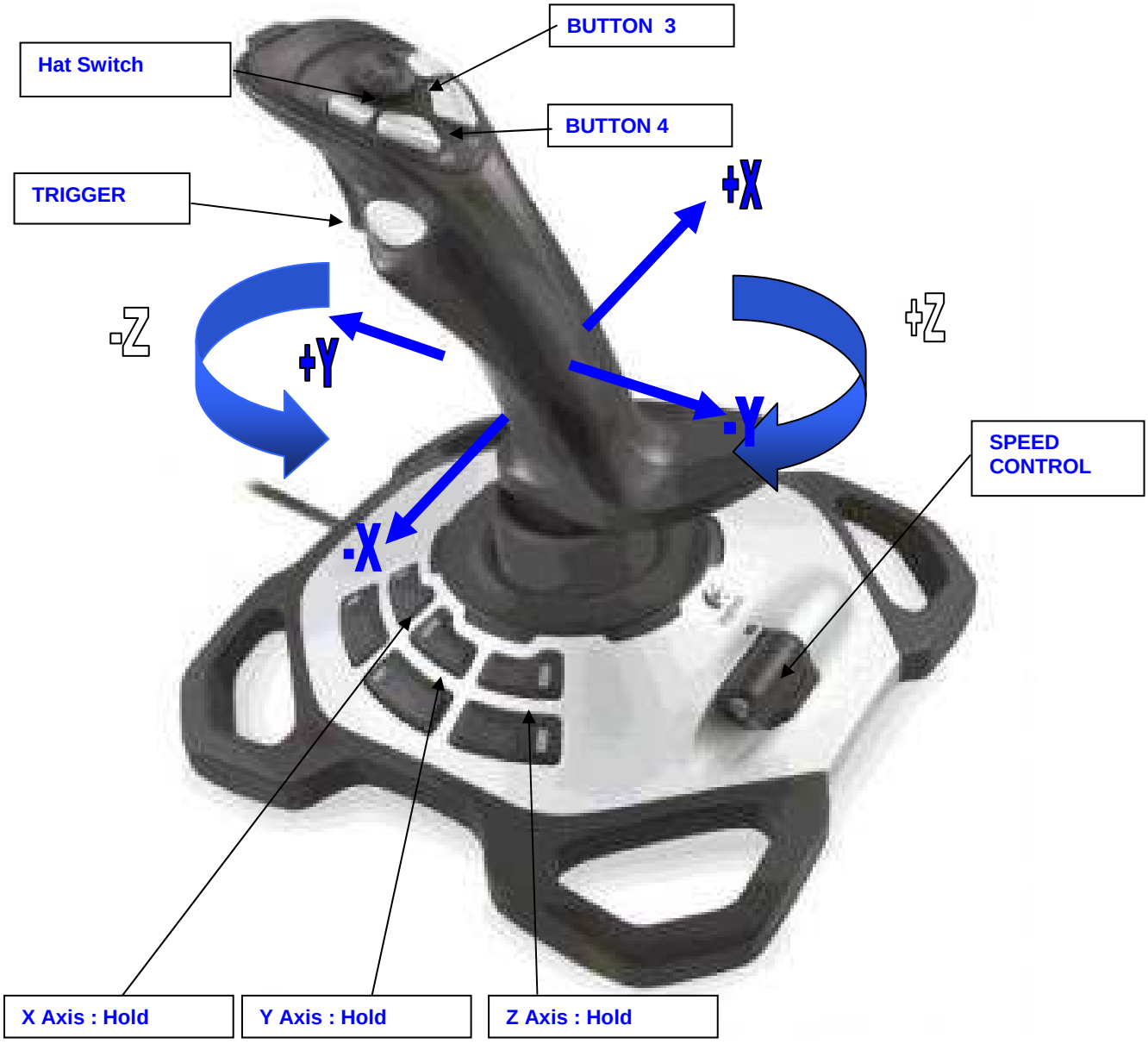
55. Machine Grid (1 .0 Introduction to CMM measurement)
Cancels all references that have been set, and returns to the machines own reference position.

56. Feature Select
Allows specific features to be selected, even when overlapping

www.EngineeringBooksPdf.com

4.0 THE JOYSTICK

The Aberlink 3D software works with a PC compatible joystick provided that the correct drivers have been installed. The CMM is supplied with a Logitech Extreme 3D pro Joystick, as shown:-



The joystick can be engaged and disengaged in the Aberlink 3D software using the following button:

www.EngineeringBooksPdf.com

4.1 Movement X, Y and Z

Moving the CMM in the X axis direction is achieved by moving the joystick to the left and right.

Moving the CMM in the Y axis direction is achieved by moving the joystick forwards and backwards.

Moving the CMM in the Z axis direction is achieved by twisting the joystick. The default setting for the joystick is clockwise drives in a Z positive direction (upwards) and anti-clockwise drives in a negative direction (downwards), although this can be altered in the machine set up.

4.2 Taking Points (at constant feed)

The joystick provides the additional function of allowing measurement points to be taken at a constant feed rate. This means that the probe does not have to be driven into the component being inspected. It can be positioned within 10mm of the desired point, and then the following buttons will produce a move at the default probing speed in the directions as detailed below. If the probe does not take a point within 20mm of the measurement direction it will retreat to its' previous position.

Button 3 - Will travel in positive Z (upwards) until the probe is triggered.

Button 4 - Will travel in negative Z (downwards) until the probe is triggered.

Button 1 - Is a hat switch. Moving the switch to the left and right moves the machine in the X- and X+ direction Moving the switch forwards and backwards moves the machine in the Y- and Y+ direction. The hat switch also allows for probing at 45˚ in the XY plane.

4.3 Indexing the Probe

Buttons 5 – 8 can be used for indexing the probe head position in place of the 4 little black arrows adjacent to the Probe Datum Button in the top left hand corner of the main software screen.
Each button will index the probe to the next datumed position:

either with a higher A value
or with a lower A value
or with a higher B value
or with a lower B value

www.EngineeringBooksPdf.com

You can press one button repeatedly to cursor through all the datumed positions, or you can use a combination of buttons to take the most direct way to get to the required position. As soon as you pause for more than 2 seconds the probe head will automatically index to that position, and you will have to wait until the index is completed to select an alternative position.

4.4 Locking the Axes

Any of the machine axes can be locked as follows:

X Axis : Hold down the shift button (button 2) and press button 5
Y Axis : Hold down the shift button (button 2) and press button 6
Z Axis : Hold down the shift button (button 2) and press button 7

Remember that if you do not hold the shift button (button 2) down then the probe will index.

Repeat the sequence to unlock each axis.

When an axis is locked the corresponding readout will appear in red.

4.5 The Speed Control

The speed control is located in the base of the joystick as shown in the figure above. When the control knob is towards the operator the speed is minimum, and away from the operator is maximum.

The default settings for this control are 10% speed at minimum and 100% at maximum, although these values can also be changed in the Machine Set Up.

4.6 The Trigger Button

The trigger button is located on the front of the joystick. This can be used for entering the machines current position into a ‘Move Via’ command.

www.EngineeringBooksPdf.com

5.0 DATUMING THE PROBE

Background

When using the Aberlink 3D software to inspect a component, the software must know the relative position and diameter of the stylus ball being used.

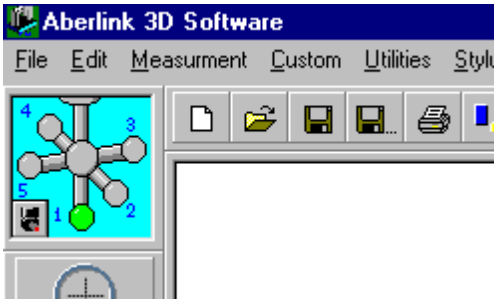
This is achieved by using the probe stylus ball to measure the reference ball, which is mounted on the granite table. As the size and position of the reference ball are known, this measurement is used to determine the size and relative position of the stylus ball. This information is stored as three position offsets X,Y and Z, and a diameter.

Obtaining this information is known as datuming the probe. The probe must be datumed every time you change a stylus, or if you have an indexible probe, each time you move the probe head to a position that has not previously been datumed.

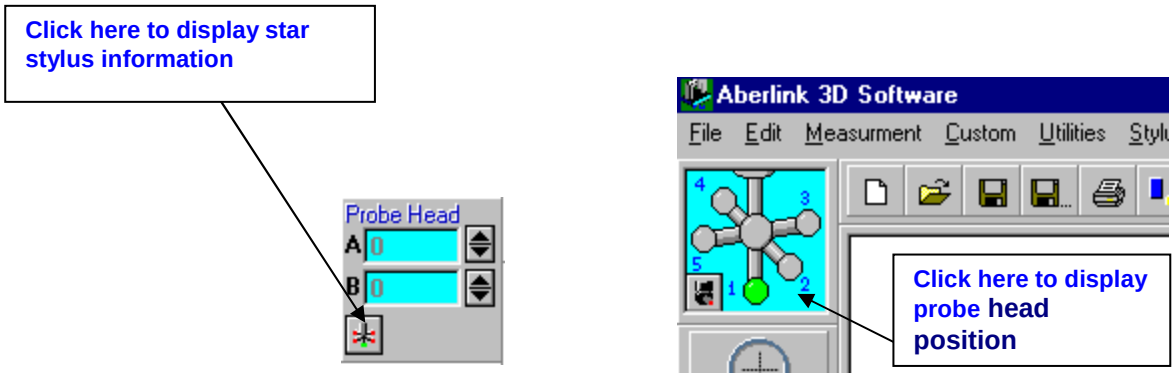
Try to datum all relevant probe positions prior to commencing your inspection. A change of styli in the middle of an inspection should be avoided, if possible.

5.1 The Probe Datumming Button

The probe datumming button in the top left hand corner of the main screen can be used to display either star stylus information, as follows:



Or if you click on the small probe head symbol, the icon will change to display probe head position (A and B axes).

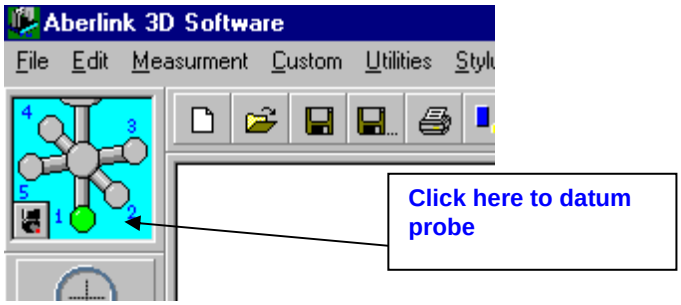


www.EngineeringBooksPdf.com

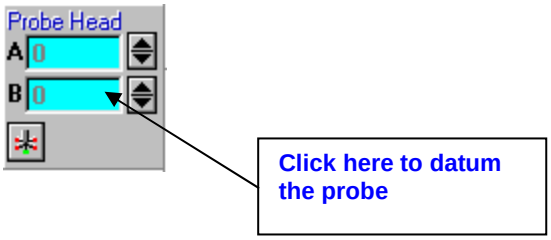
5.2 Datumming the Probe

To bring up the Probe status Window click on the Probe Datumming button:-

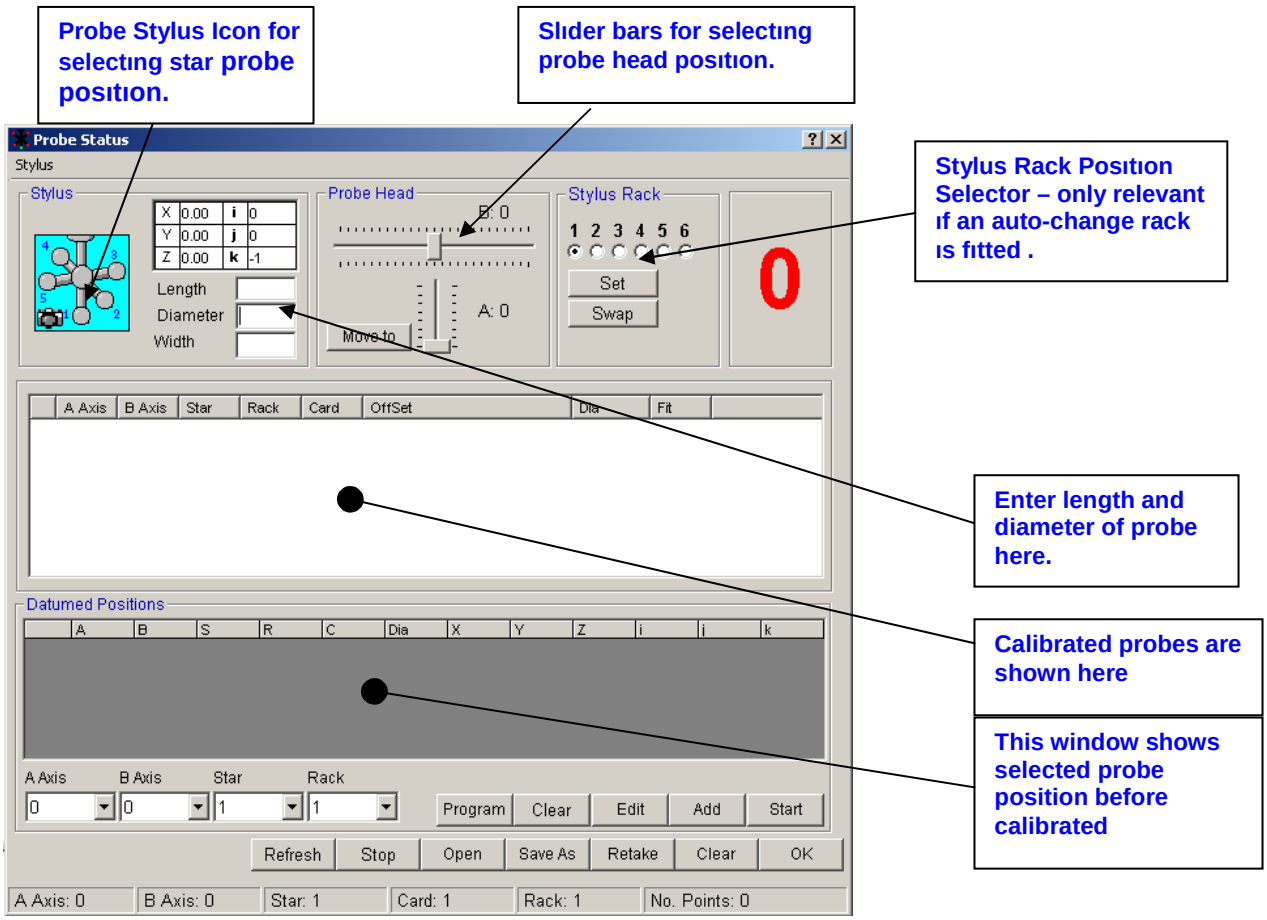
Either on the light blue area when it is displaying star stylus information:



Or on the green background when it is displaying probe head positions:



This will bring up the probe status window:



www.EngineeringBooksPdf.com

The top of the window is split into 3 sections which you can use to select which stylus position you wish to datum:

The Probe Stylus Icon is used to select which ball is being datummed if a star stylus is used.

The 2 slider bars are used to select the probe head position A and B axis (only relevant for an indexible probe). The horizontal bar at the top selects B axis position and the vertical bar at the bottom selects A axis position.

The Stylus Rack Position selector (1-6), which is only relevant if an auto change rack is fitted to a CNC machine and will default to position 1.

If you are using a star stylus you will need to click on the relevant ball position within the Probe Stylus Icon.

If you are using an ordinary straight stylus (or even position 1 on a star stylus) there is no need to click on the Probe Stylus Icon, as it will automatically default to position 1 (straight down).

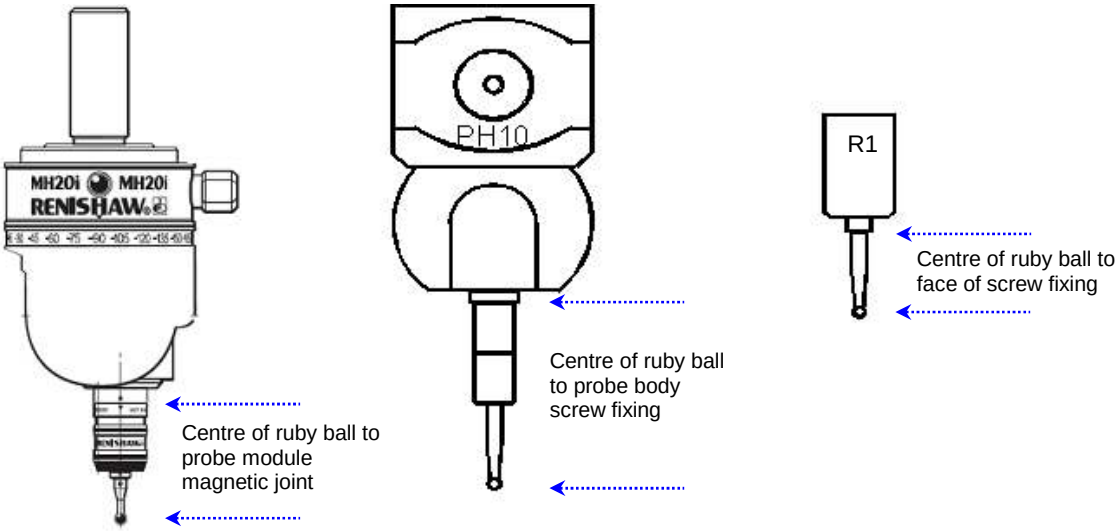
If you are using an indexible probe head (eg. PH10T) use the 2 slider bars to select the probe head position that you want to datum. The horizontal bar at the top selects B axis position and the vertical bar at the bottom selects A axis position.
The Rack Position selector is not relevant for a manual machine – it is only used if an auto-change rack is fitted to a CNC machine and will always default to position 1.
We also need to enter approximate values for the length and diameter of the stylus being used. Roughly measure (+/-1mm) the length protruding below the probe head:

The method for determining the probe length in the probe status window varies according to probe type. The table below lists the more common probe types and the method of determining the correct probe length.

Probe Type	Probe Length
MIP	Stylus length - from centre of ruby ball to face of screw fixing
PH50	Stylus length - from centre of ruby ball to face of screw fixing
R1/PH6	Stylus length - from centre of ruby ball to face of screw fixing
TP8	Stylus length - from centre of ruby ball to face of screw fixing
RTP	Stylus length - from centre of ruby ball to face of screw fixing
PH9/PH10	Stylus length plus complete module length - from centre of ruby ball to probe body screw fixing
MH20i	Stylus length plus probe module - from centre of ruby ball to probe module magnetic joint (usually the probe length plus 20mm)

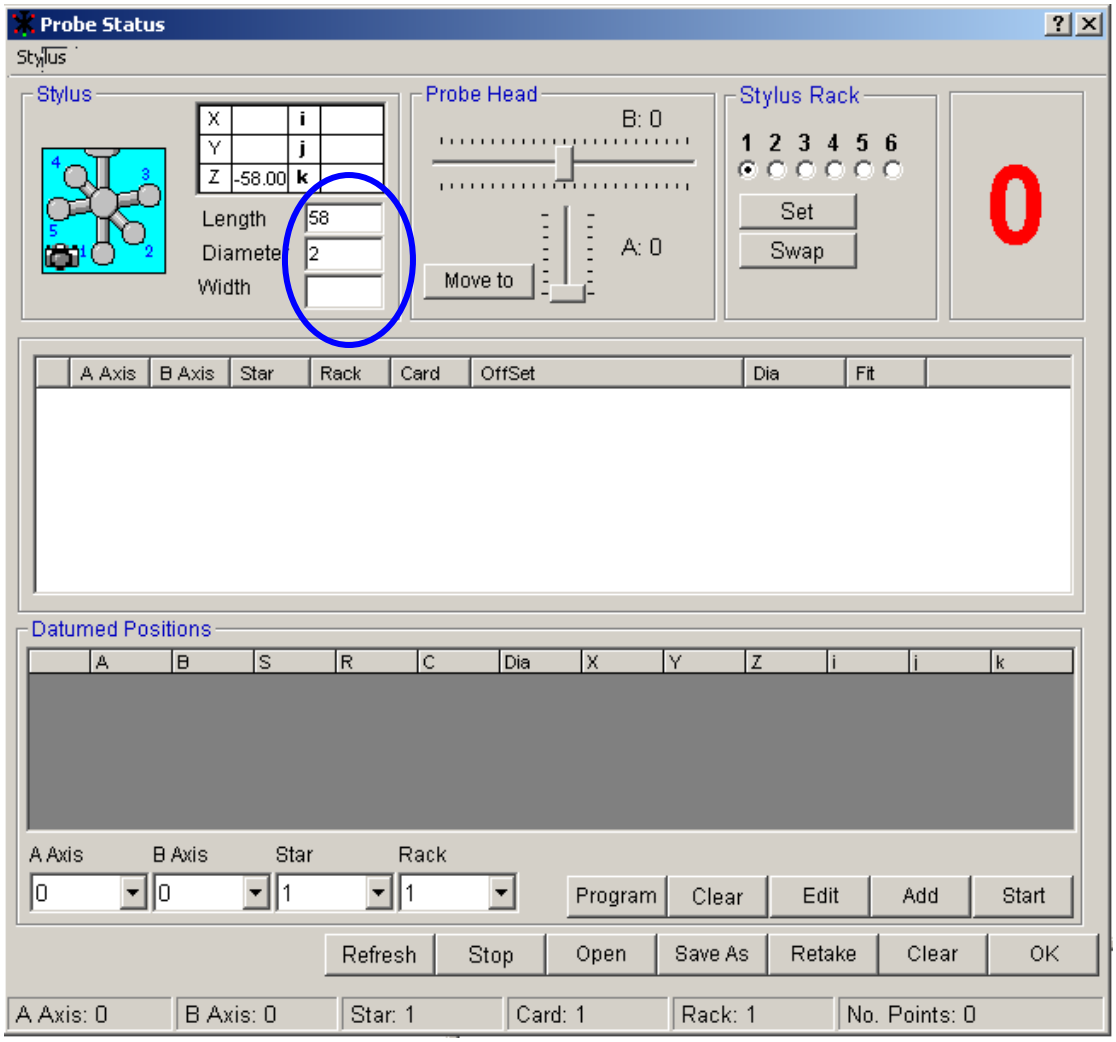
www.EngineeringBooksPdf.com

Some examples are shown below.



www.EngineeringBooksPdf.com

Enter this length and the ball diameter at the top of the Probe Status Window as show below.

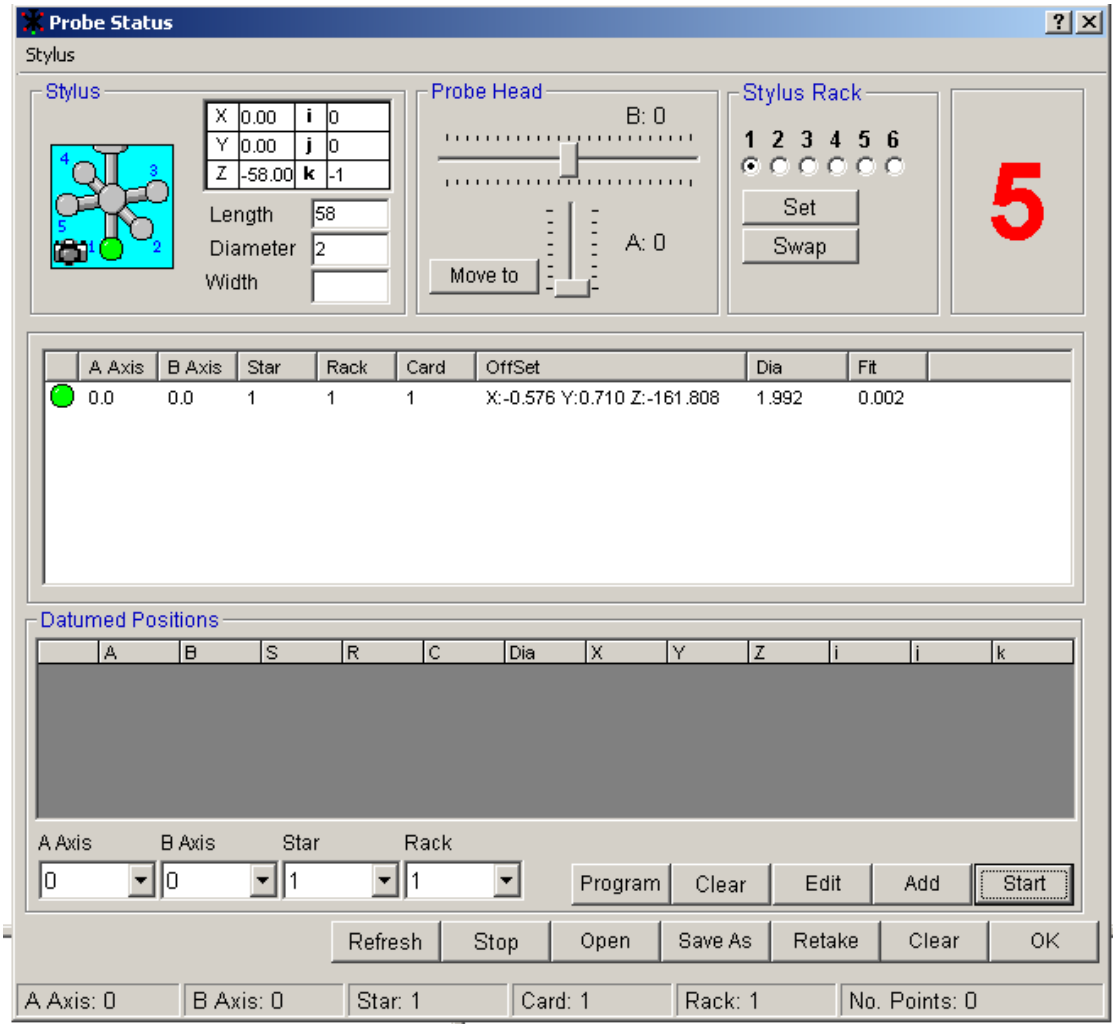


If the offsets previously measured relate to the current stylus and you wish to keep them click on ‘No’. The new position selected will then be added to the original offsets.

Note: This prompt will only appear after the first position has been datummed, and will not be repeated for subsequent datumming positions unless the Probe Status Window is closed and then re-opened.

When a probe position has been successfully datummed the information about the probe will be transferred to the central white area:

www.EngineeringBooksPdf.com



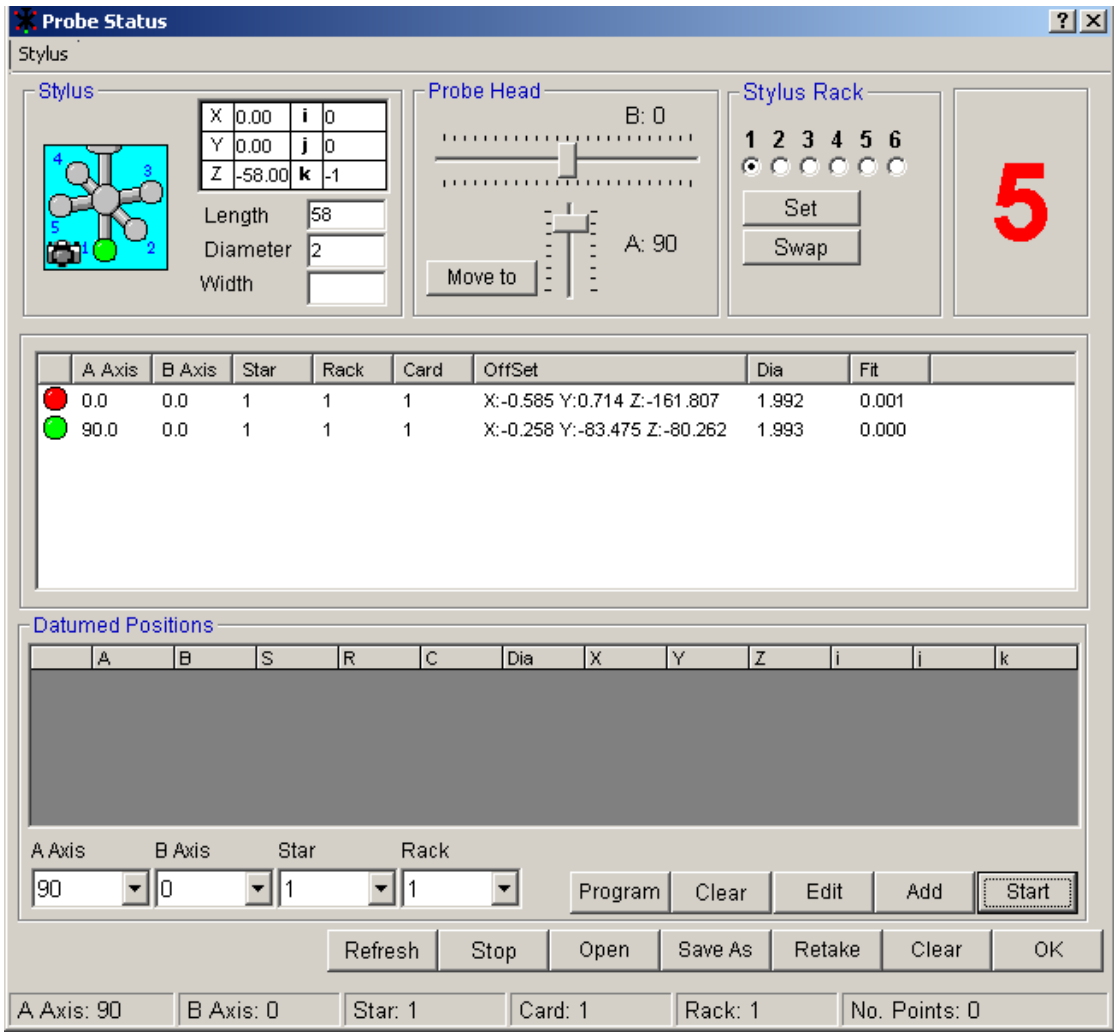
This includes the probe head position, star position, rack position, X,Y,Z and diameter offsets and fit for the points taken.

This information will be preceded with a green dot to show that it is the current stylus position. Any other positions that have been datummed will be preceded with a red dot to show that this position has been datummed but is not current.

A yellow dot denotes that the points fit is too high and the calibration is not ‘safe’. This can be caused by a loose stylus or a dirty ball etc. Check for the cause and repeat again.

To Repeat this process in a different datuming position click on the sidebar slider, drag to desired probe head position (say A 90 B 0) click on the **‘Move to’** button. Probe head will automatically index to that qualified position. Click on the ‘**Add’**button. The probe selected will now be entered into the grey area at the bottom of the window **.** Click on the **‘start’** button. The machine will now start to automatically measure the probe in its current datuming position.

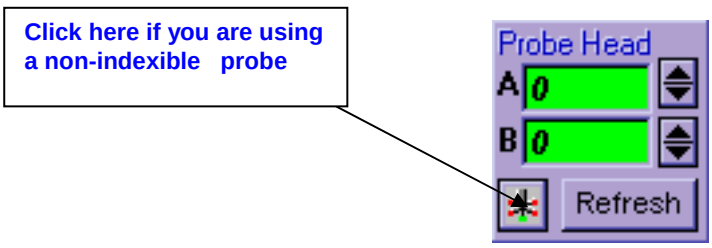
www.EngineeringBooksPdf.com



5.3 Using a non-indexible probe

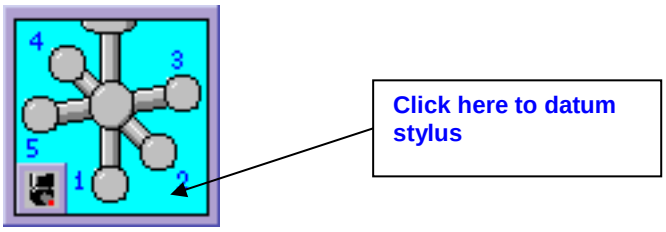
With a non-indexible probe the maximum number of position that can be datummed is 5, when using a star stylus. Therefore this can be represented using an icon (rather than the grid for an indexible probe).

If the probe datumming button is showing an A and B value (which relates to a indexible probe), click on the small non-indexible probe icon in the bottom left hand corner of the button:

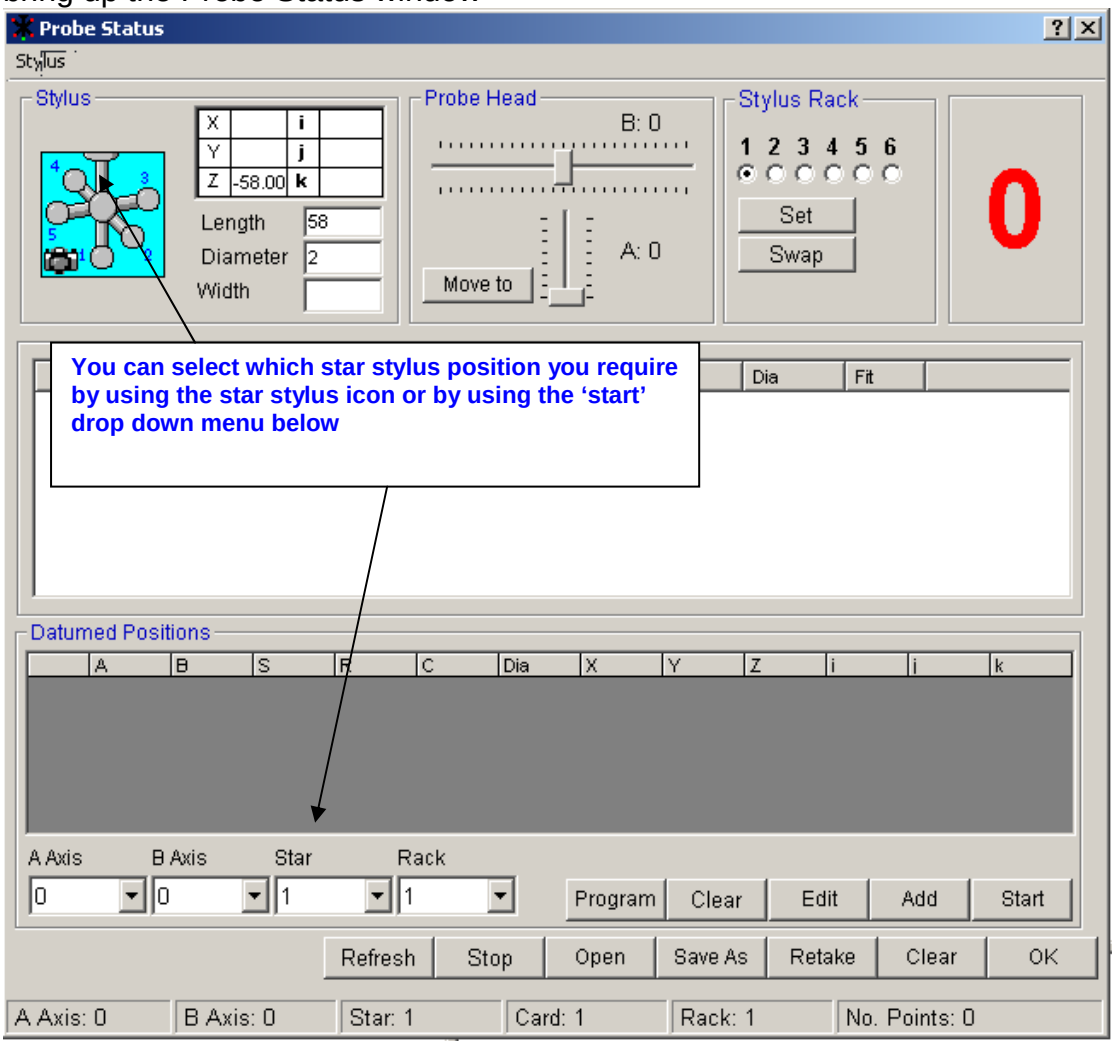


www.EngineeringBooksPdf.com

The appearance of the button will now change to show a star stylus icon:



Now to datum the probe click on the light blue background area. This will bring up the Probe Status window



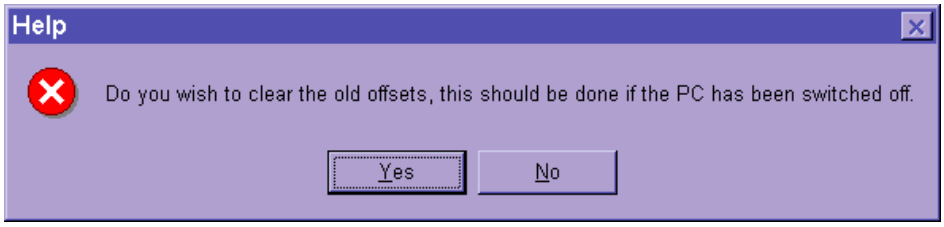
The star sylus icon is replicated at the top of the window. Position 1 represents a straight stylus or the ball in the straight down position of a star stylus. The other positions are purely for a star stylus with position 2 pointing towards the front of the machine, position 3 to the right, position 4 to the back and position 5 to the left.

When using a star stylus you can manually datum position 1 to 5 by taking points on the ref sphere using the joystick.
Or you can automatically datum position 1 to 5 by using the software.

www.EngineeringBooksPdf.com

5.4 Manually datum star stylus

Pick stylus position 1 in the star stylus window (it will now turn **red**) using the joystick take 5 points evenly spread around the reference ball, using the ruby ball in position 1.
When it has completed the 5th point at this position, the following prompt will appear
If the offsets previously measured relate to the previous stylus, click on 'Yes'.



These will then be erased, and only the offsets for the new position selected will be valid.

If the offsets previously measured relate to the current stylus and you wish to keep them click on 'No'. The new position selected will then be added to the original offsets.

The ball position selected will now turn **green**, showing that it has been datummed and is current.

> *Note : Every time the stylus is changed you should select 'Yes' for the above question. If the stylus has not been changed, and you simply want to add another position (on a star stylus) to those previously datummed then select 'No'.*

Now for example to datum the ball in position 2, click on its representation so that it turns **red.** Using the joystick take 5 points evenly spread around the reference ball, using the ruby ball in position 2.

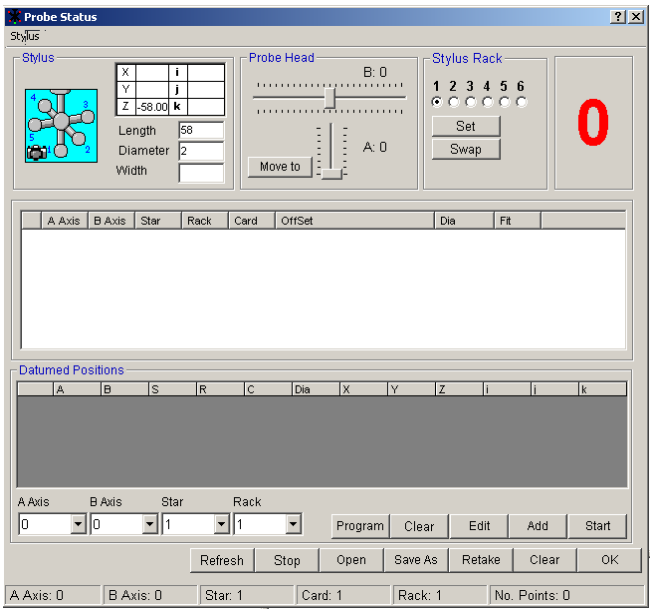
After 5 points the ball in position 2 will now turn green, showing that it has been datummed and is current, and any previously datummed positions will turn **red**, showing that they have been datummed but aren't current.

When you have finished datumming all required positions, the last position datummed will be coloured **green** (current), and the other positions datummed will be **red** (datummed but not current), with the positions not datummed remaining **grey.**
Click 'OK' and the software will return to the main screen, with the updated icon being replicated in the Probe Datum button.

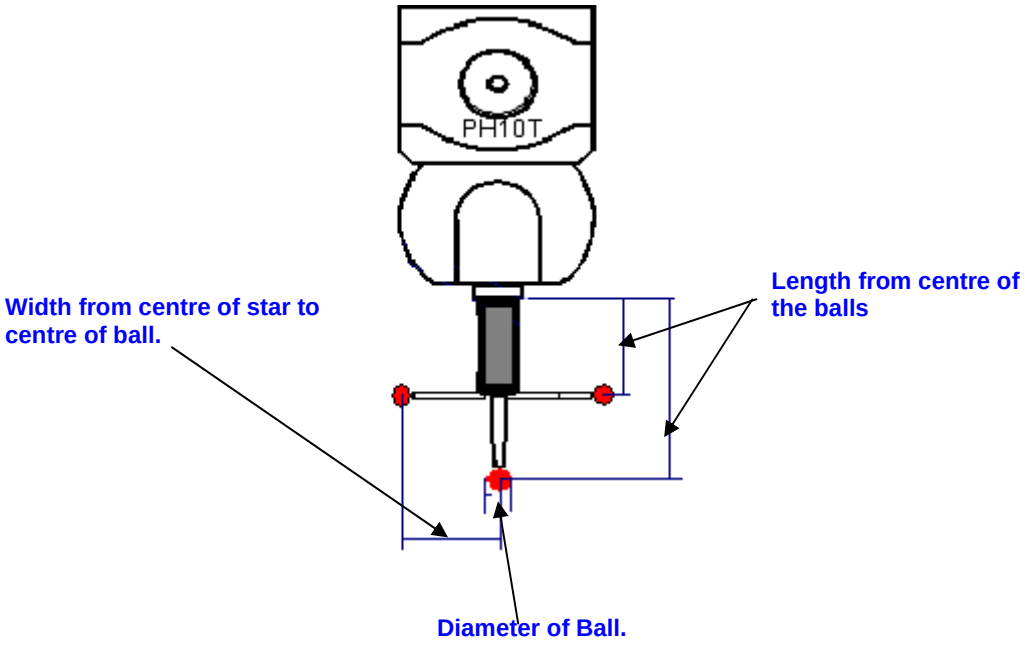
5.4 Automatically datum star stylus

1. Pick stylus position 1 using the star stylus icon or by using the 'start' drop down menu below:-

www.EngineeringBooksPdf.com



2. Enter the length of stylus 1 in the ‘length box’, then enter the diameter of the ball in the ‘Diameter box’

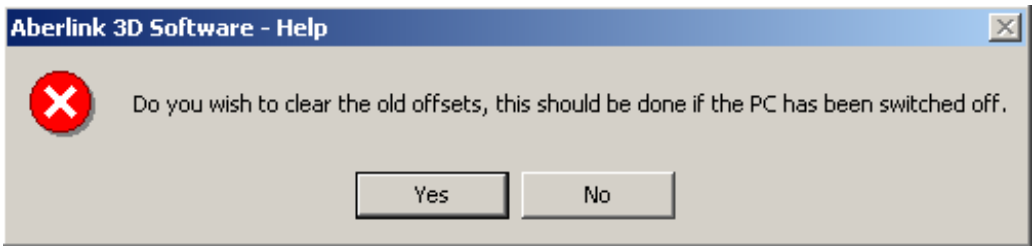


Now **click on the ‘Add’** button at the bottom of the window. The probe position selected will now be entered into the grey area at the bottom of the window:

Here we have selected Probe Head position A0 B0, star position 1, (and also by default rack position 1).

www.EngineeringBooksPdf.com

Click on 'start'. The machine will now start to automatically datum the probe in all of the positions selected. When it has completed the 5th point at the first position, the following prompt will appear:



If the offsets previously measured relate to the previous stylus, click on 'Yes'. These will then be erased, and only the offsets for the new position selected will be valid.

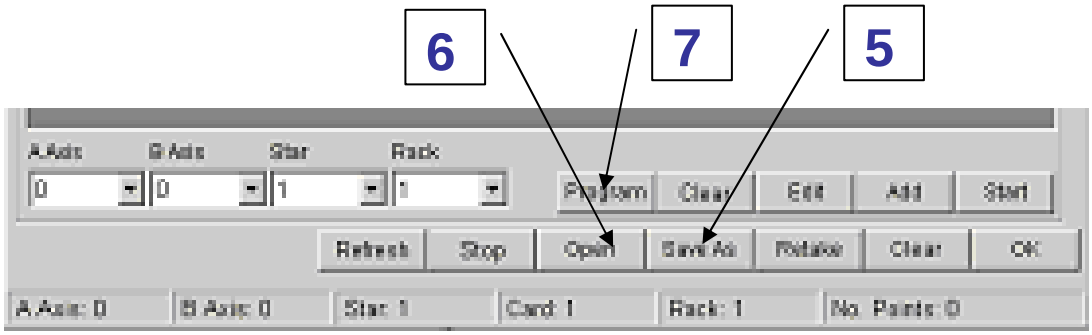
If the position previously measured relate to the current star stylus you need to click on 'No'.
The ball position selected will now turn **green**, showing that it has been datummed and is current.

Note: This prompt will only appear after the first position has been datummed, and will not be repeated for subsequent datumming positions unless the Probe Status Window is closed and then re-opened.
When a probe position has been successfully datummed the information about the probe will be transferred to the central white area:

Repeat this process for star stylus 2 to 5 (remembering to change the length in the 'Length Box')

When the prompt box appears remember to click on 'NO' as this will add the stylus to the calibrated stylus box.

5.5 Saving Probe list



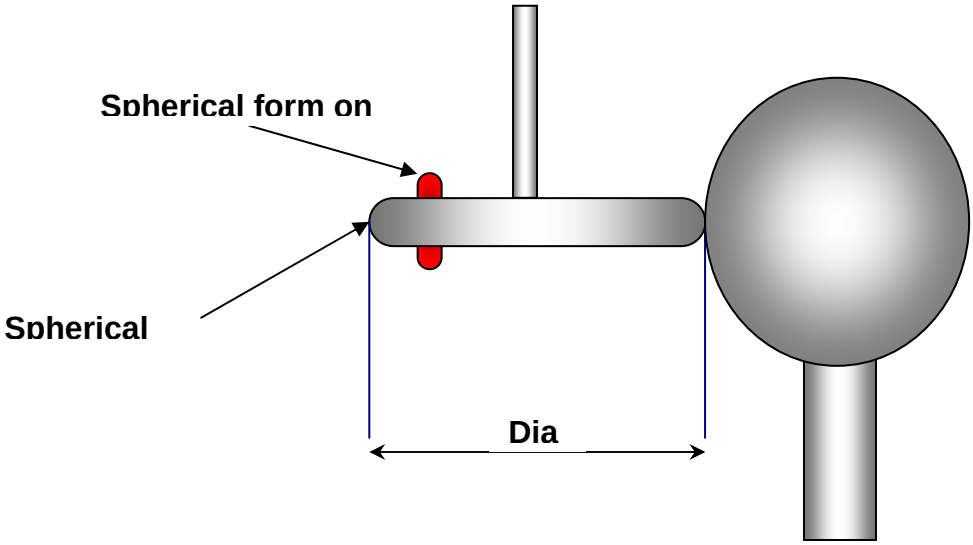
5. By clicking on the 'Save As' button you can save the current calibrated probe list to a file.

6. By clicking on 'Open ' button you can load saved probe files

www.EngineeringBooksPdf.com

7. By clicking on the ‘Program’ button the software looks at the current measurement program and will list the probe needed to measure that part.

5.6 Datumming using a Disc Stylus



1. Position the disc half way up the reference sphere ,Call up the probe status window and select offset 1 on the star probe graphic (the ball will go **Red**) for redatuming and take five points around the outside diameter of the ref sphere varying the height to cover the spherical form along the edge of the disc
2. Measure the length f the stud using a micrometer or vernier and note. The length will also be its diameter.
3. Select offset 2 from the star probe graphic ,the ball will go **red.** Take about 10 points using the spherical form on the bottom of the stud (only a small area on top of the ball is accessible). This will give a position of the sphere,s center relative to the bottom of the Z quill.
4. Edit the ball diameter on offset 2 so its matches the length previously measured. Also you need edit the diameter of offset 1 to match the actual disc diameter.

Click ‘OK’ at the bottom of the probe status window using the star probe graphic it is now possible to toggle between the two offsets by clicking on the relevant ball position.

5.7 Tool changing rack

If you have a Renishaw MCR 20 probe rack fitted to your machine it is possible to change modules automatically from within a program saving you

www.EngineeringBooksPdf.com

the trouble of having to datum the probe every time you change to a different probe.

The probe rack can hold up to six modules each with a different stylus set up which can include star stylus, disk stylus & long reach stylus.

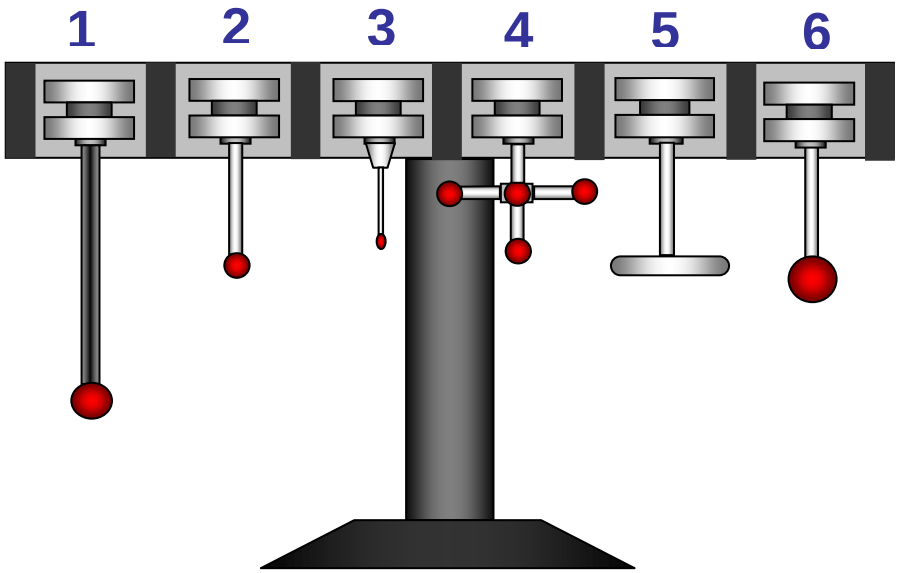
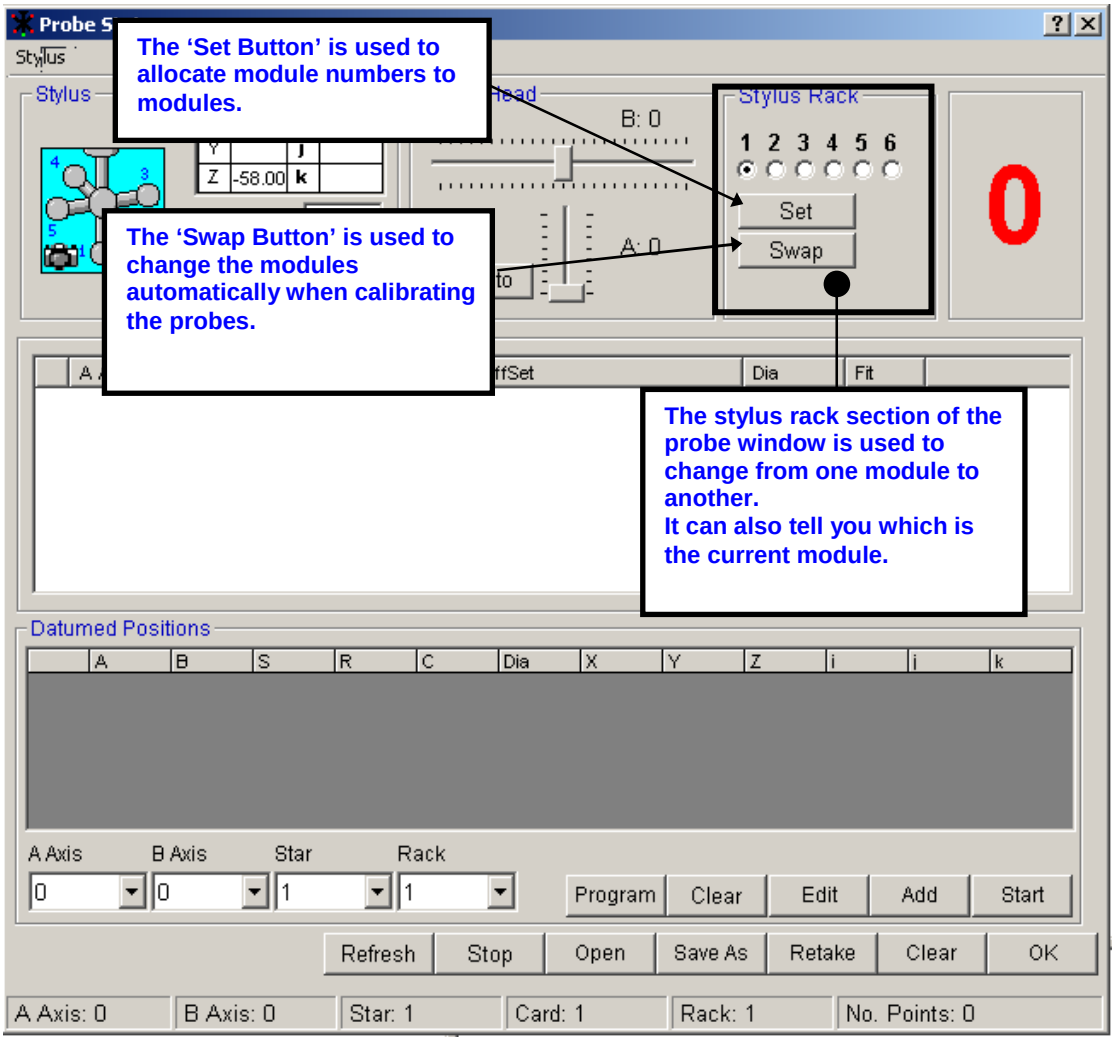


Fig 5.7-1 Modules in tool change rack



www.EngineeringBooksPdf.com

Fig5.7-2 Modules selection box in probe status

The six black buttons in the stylus rack selection window corresponds to the six module positions in the tool rack.
If button one is black (•) this means 'module one' has been selected from the tool changer. If you were to click on button number two it will go black (•) and the CMM will automatically change from module one to two.

When the CMM changes modules it follows a predefined probe change routine, which moves the probe head along safe path between the part and the change rack.

The 'Set Button'
This button tells the CMM which module is currently being used to measure parts.
If you click on button six (•) and then click on the set button this will allocate position six to the module in the probe head.

Note:- There is no feedback between the CMM and the change rack so if you have a problem and have to restart the software it is possible for the machine to think it has the wrong module in the probe head.
This where the 'swap button' is useful in allocating the correct module to the probe head.

The 'Swap Button'.
This button is used to change modules when calibrating probes.
After calibrating all the A & B angles for module one, click on the number two button in the stylus rack window and this will change the module automatically from one to two. You can now calibrating all the A & B angles for module two.

www.EngineeringBooksPdf.com

6.0 ALIGNING THE AXES OF A COMPONENT

6.0.1 Introduction

When using the CMM to inspect a component, it is not necessary to physically align the component to the axes of the machine. By measuring specific features on the component and setting them as references, you will be able to define the orientation of the axes and position of the component.

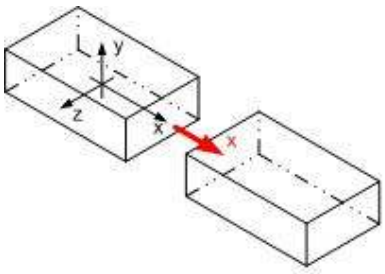
This is important when measuring 2 dimensional shapes ie. lines and circles, or points, as the software will project the measurement points into a plane which must be fully defined. Also the alignment of the component must be properly defined in order to produce meaningful horizontal and vertical dimensions on the screen. Grasping this concept is fundamental to utilising these features to their full potential, and hence it is important to understand the principles described in this chapter before continuing with the rest of the manual.

6.0.2 Example of traditional part alignment using a surface table, straight edge & pins.

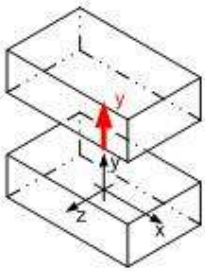
When you place a part on a surface table you are aligning the bottom face of that part, true to the top of the surface table, you can still move the part about in the X-Y plane & rotated about the Z axis but you can not move it up and down in the Z axis or rotated around the X or Y axis. This is called a ***primary alignment***.

If you then push the part up to a straight edge fixed to the surface table this will stop any rotation about the Z axis as well any movement along the Y axis. Now you can only slide the part along the X axis.
This is called a ***secondary alignment.***

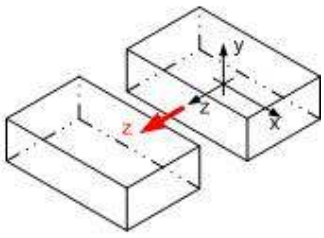
If you now slide the part along the X axis up to a pin in the surface table, the part is now fixed in all six degrees of freedom. It cannot move or be rotated in X Y or Z.



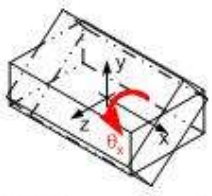
Linear in x-direction



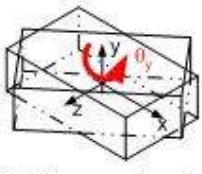
Linear in y-direction



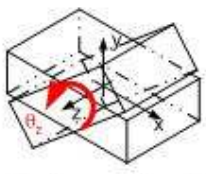
Linear in z-direction



Rotation around x-axis



Rotation around y-axis



Rotation around z-axis

www.EngineeringBooksPdf.com

6.0.3 Example of alignment using Aberlink 3D

When you place the component on the table of the machine it has 6 degrees of freedom which will need to be defined, in order to fully define the alignment and position of the component. Namely these are translation along, and rotation about each of the 3 axis.

Depending on what measurements you wish to perform, not all of these degrees of freedom will necessarily have to be defined. Take for example a cube with holes in its top, front and side faces:

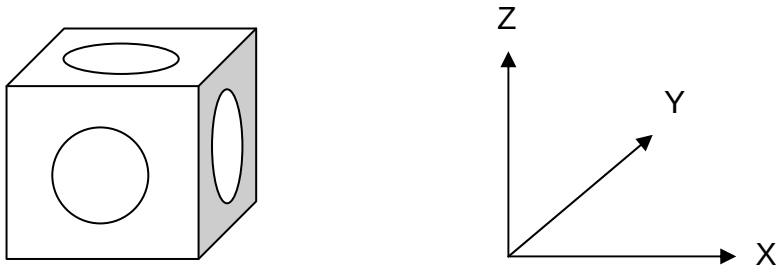


Fig. 6.0-1 A cube with holes in its top, front and side faces

Measure a plane on the top face of the cube and set it as a reference. This defines the XY plane for the cube. In effect it defines 3 of the degrees of freedom, which are rotations about X and Y, and also translation in Z.

Now measure the hole in the top face of the cube as a circle. The software will automatically determine that the hole is in an XY plane by looking at the direction of motion of the probe, and this plane will be shown in the diagonal corners of the graphics window. Note that the X,Y,Z and D values for the hole will be shown in red, denoting that the XY plane and Z position have been defined, and that the diameter for the hole has been correctly calculated.

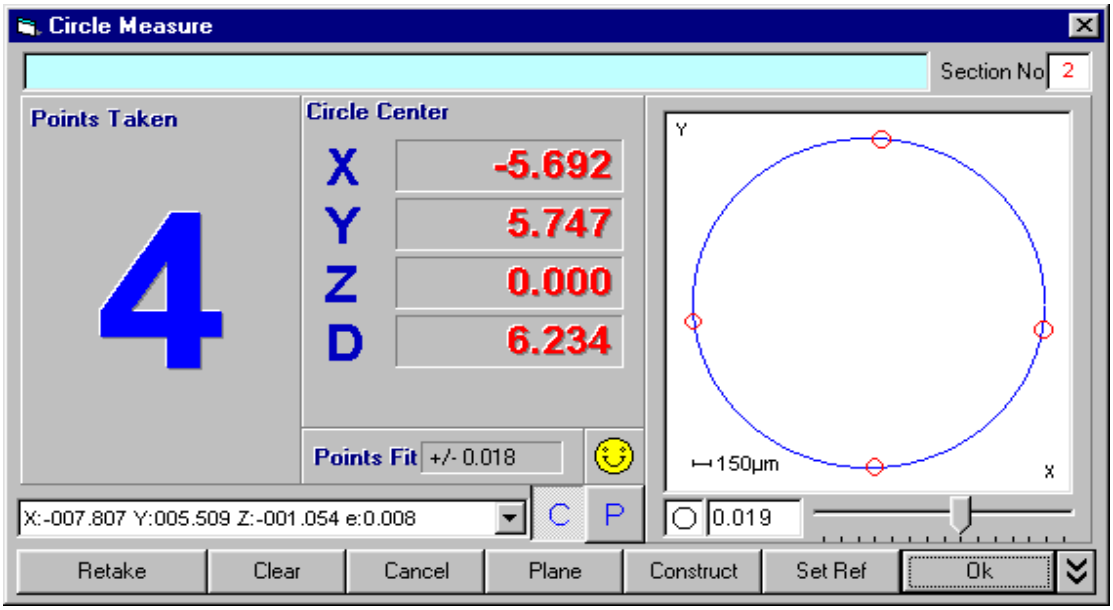


Fig. 6.0-2 The Circle Measure window

www.EngineeringBooksPdf.com

Clicking on OK will return you to the Main Screen:

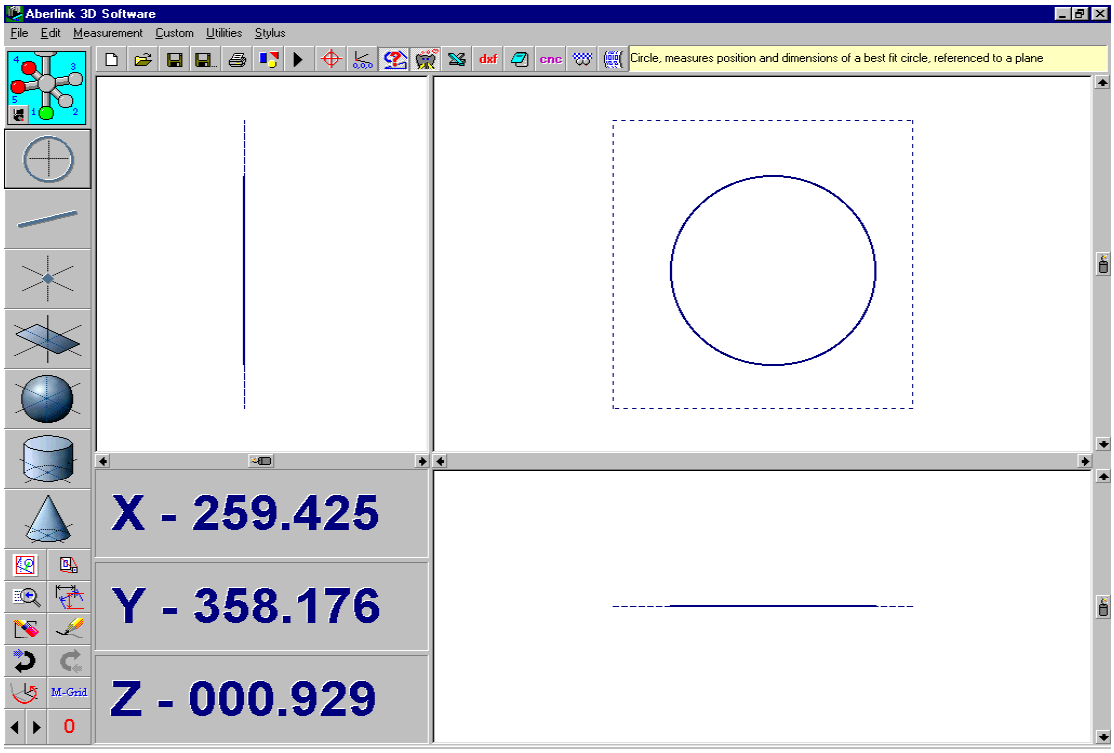


Fig. 6.0-3 The Main Screen showing a measured plane and circle

Now measure the front, back and side faces of the cube as lines. Again the software will automatically determine that the lines are in an XY plane and the I,J,K and L values for the lines (when the lines have been fully extended) will be shown in red.

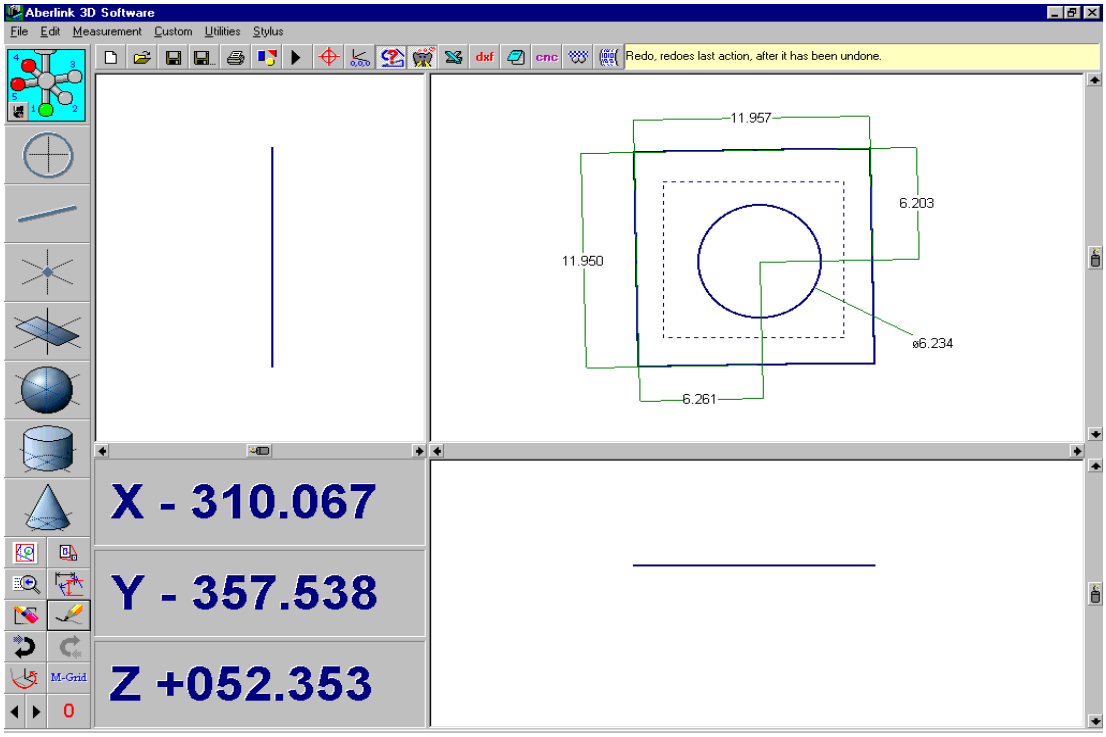


Fig. 6.0-4 The Main Screen showing the top face of the cube fully dimensioned

www.EngineeringBooksPdf.com

If these were the only measurements that you wished to perform on the cube, then there would be no requirement to further define the orientation and position of the cube. If however, you now measure the hole on the front face of the cube, the software will automatically determine that the hole is in an XZ plane by looking at the direction of motion of the probe. However, the X,Y,Z and D values for the hole will be shown in a light grey colour, rather than the usual red, by way of a warning that the XZ plane, and the position in Y have not been properly defined.

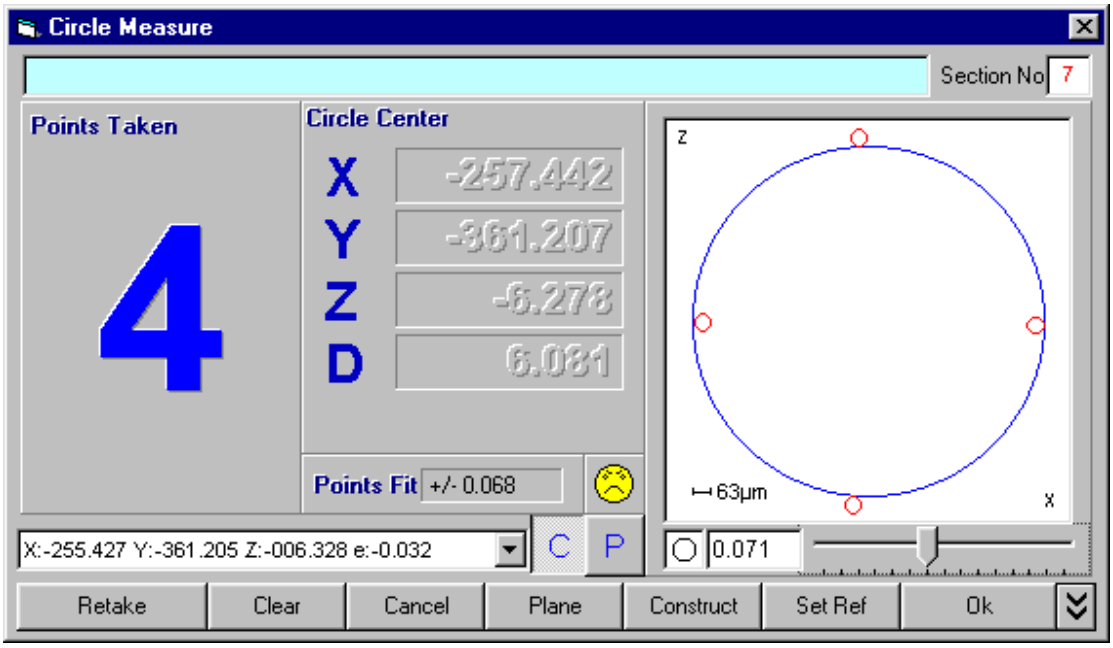


Fig. 6.0-5 A Circle Measure window showing a circle that has been measured in the XZ plane, which has not been defined

The software will know that the XZ plane of the component is perpendicular to the XY plane, but its rotation about the Z axis and position along the Y axis have not yet been defined. The information shown in grey has therefore been calculated using the machine's X axis to define the rotation about Z, and the average Y value of the measurement points taken has been used to define the circle's position in Y. If the cube is not aligned to the machine's axes, this will lead to errors in both the position and diameter of the hole.

www.EngineeringBooksPdf.com

Click on OK to return to the main screen anyway.

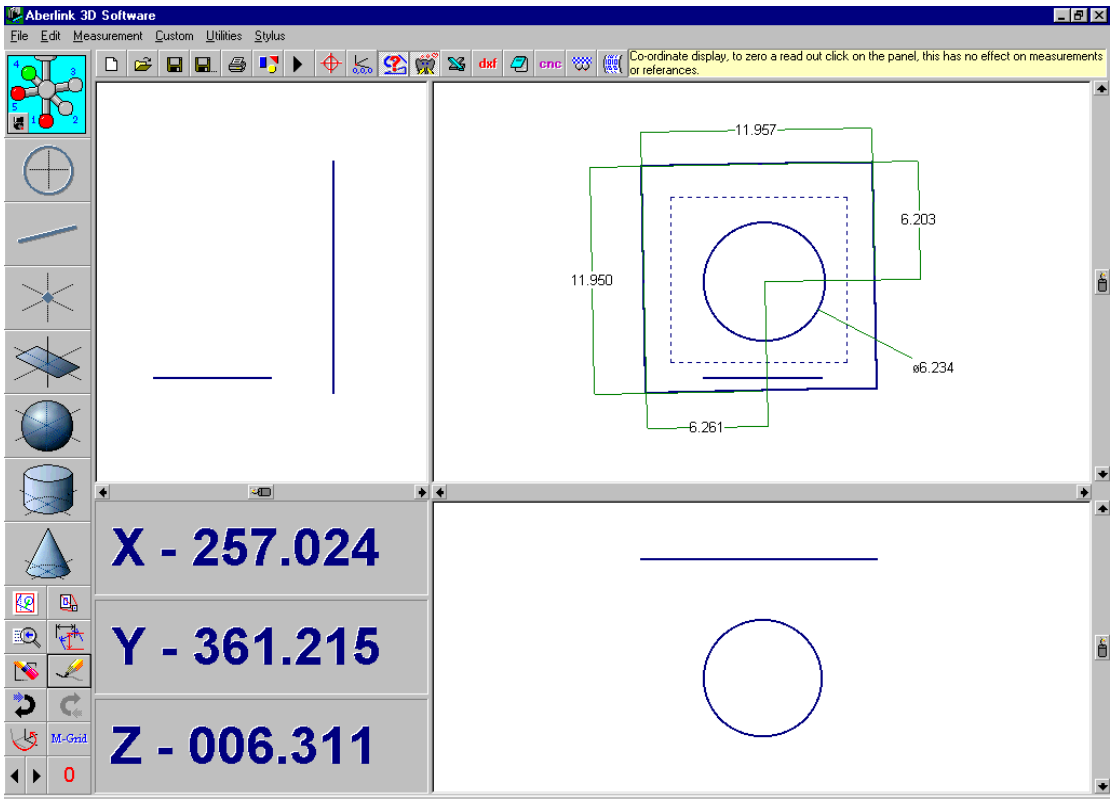


Fig. 6.0-6 The Main Screen showing the hole in the front face in an undefined plane

You will notice that the hole has not been projected into the front face of the cube. Now recall the line unit representing the front face of the block, and set it as a reference:

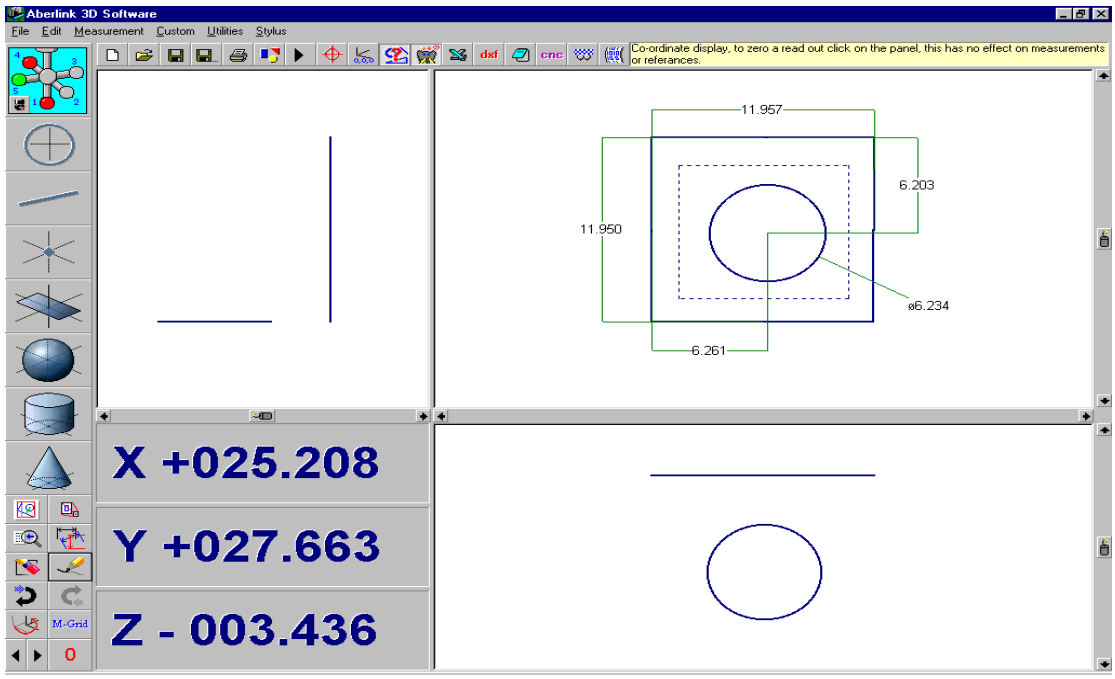


Fig. 5.0-7 The Main Screen showing the hole in the front face now in a defined plane

www.EngineeringBooksPdf.com

By setting this line as a reference we now define two further degrees of freedom, namely rotation about the Z axis, and translation along the Y axis. Hence the orientation and position of the cube's XZ plane has now been fully defined and the graphical representation will now be aligned to the front face of the block. The circle that we measured in the front face is now projected into this plane. If you recall the circle unit you will see that the position and diameter of the hole will probably have changed slightly, but now will be displayed in red.

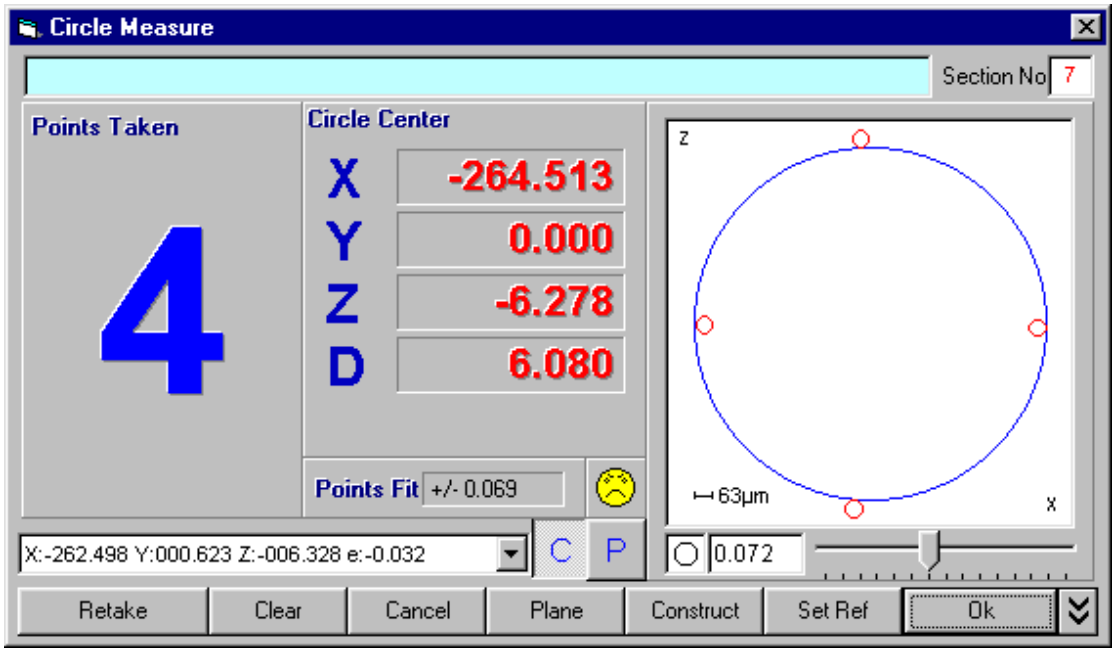


Fig. 6.0-8 A hole in the XZ plane, which has now been defined

Now measure the hole on the side face of the block. Again the software will automatically determine the plane of the hole, this time a YZ plane. In this instance, however, as we have already defined the XY and XZ planes, the orientation of the YZ plane is also defined, but the sixth degree of freedom which has not yet been set, is it's position along the X axis. The software will therefore be able to calculate the diameter and YZ positions of the hole correctly, but will still have to use the average X values of the measurement points taken to determine the circle's position in X. In this case the Y,Z and D values for the hole will be shown in red, but the X value will be in light grey.

www.EngineeringBooksPdf.com

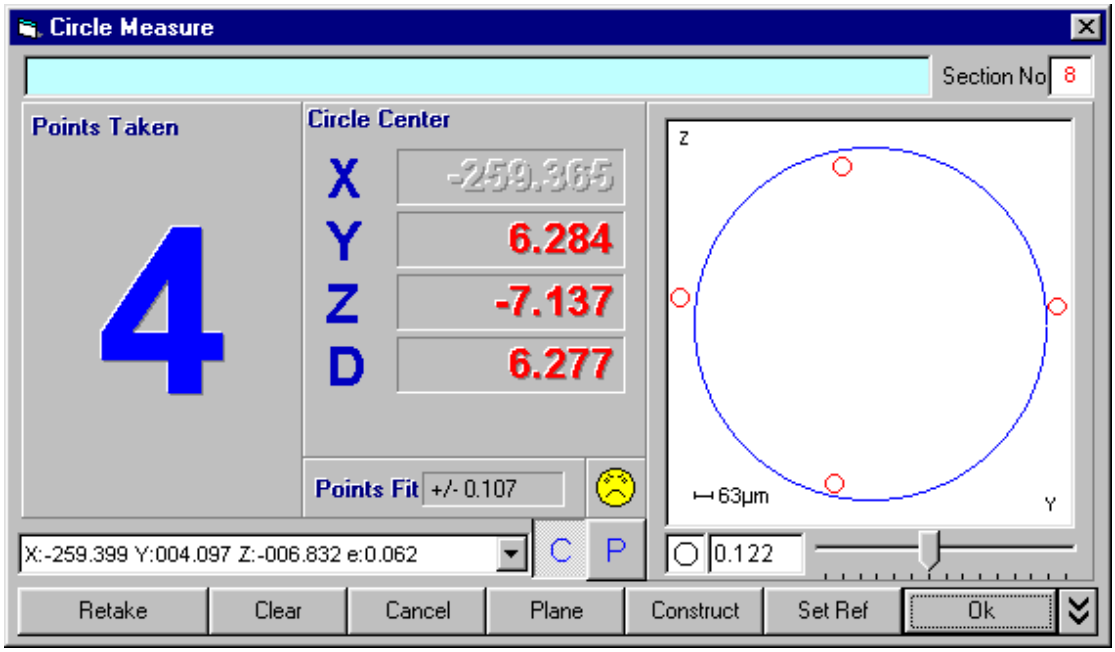


Fig. 6.0-9 A Circle Measure window showing a circle that has been measured in the YZ plane, which has been defined, but which does not have a position

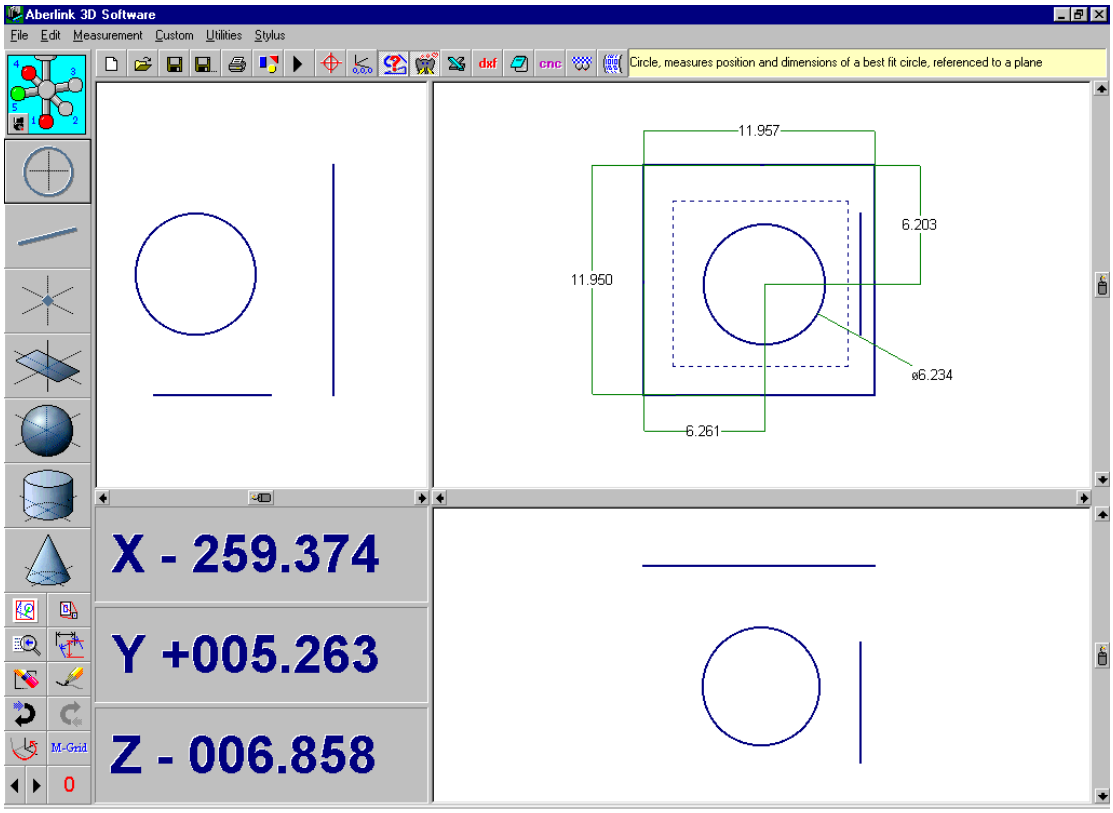


Fig. 6.0-10 showing the hole in the side face with its X position not defined

Now take a point measurement on the side of the block, adjacent to the hole, and set it as a reference. This point now defines the sixth degree of freedom, and the YZ plane of the block will now be positioned at this X value.

www.EngineeringBooksPdf.com

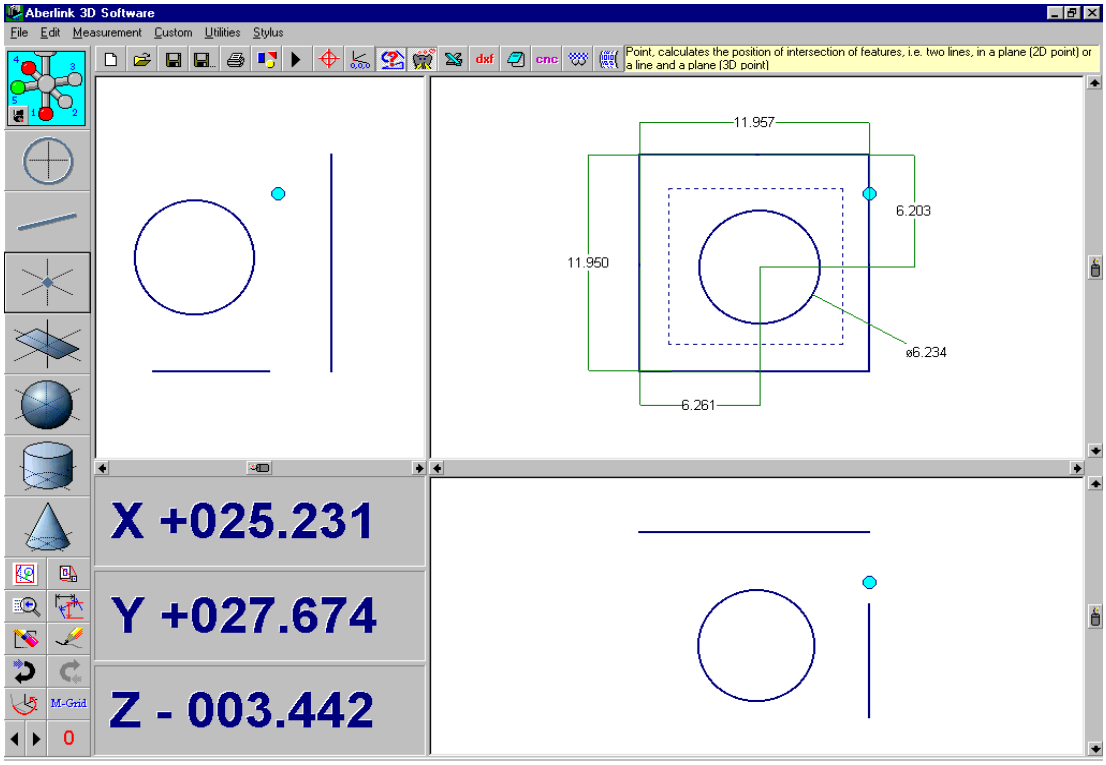


Fig. 6.0-11 The Main Screen showing the cube and holes fully defined

The circle that we measured in the side face is now projected into this plane, and if you recall the circle unit you will see that the X position of the hole is now also displayed in red.

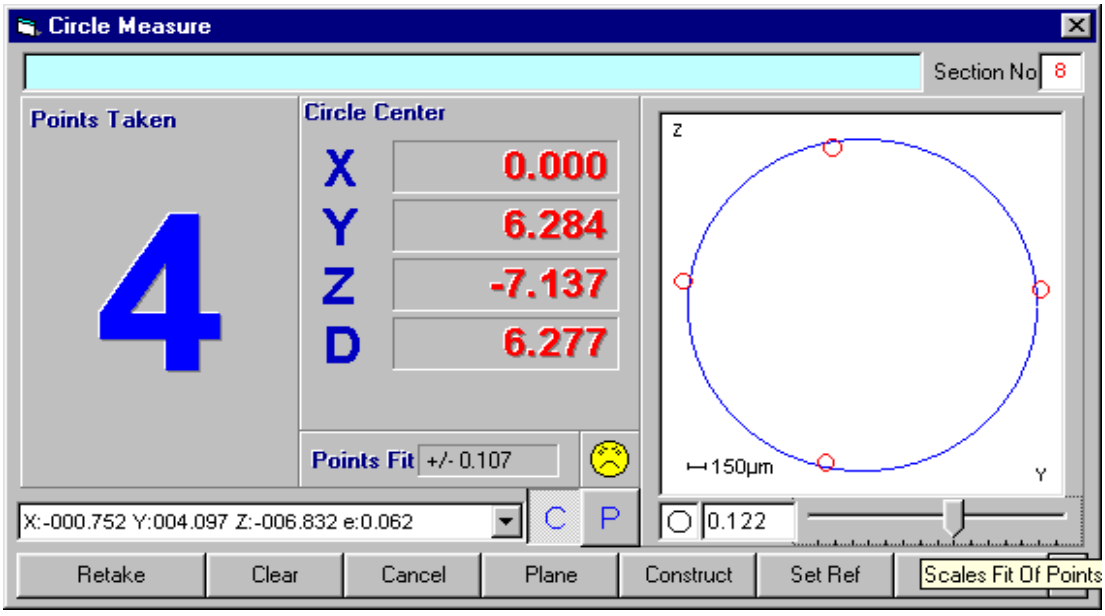


Fig. 5.0-12 The Hole in the YZ plane, now fully defined

This final reference point could also have been set by recalling the line representing the right hand edge of the cube and setting it as a reference. In this case the cube would then be aligned to the right hand edge not the front edge, as the latter reference overrides the initial one. In this case the front

www.EngineeringBooksPdf.com

edge is still to be used to define the 6th degree of freedom, (position in Y), and is treated as a point at it's mid-point.

In summary the six degrees of freedom for a component are usually fully defined by referencing a plane, a line and a point. Different features when referenced will define various degrees of freedom, and this is more fully discussed in appendix 1.

It is worth noting before finishing this chapter, that cylinders and cones when referenced will define four degrees of freedom. Take a cylinder that has been referenced to define the X axis of the component.

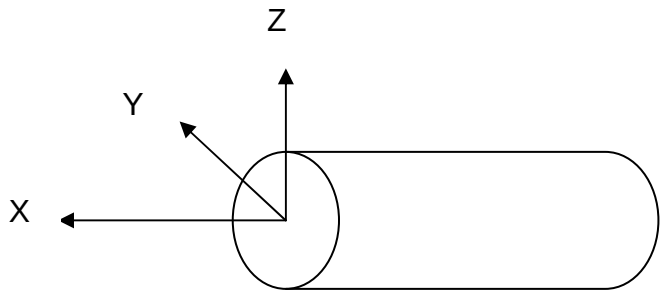
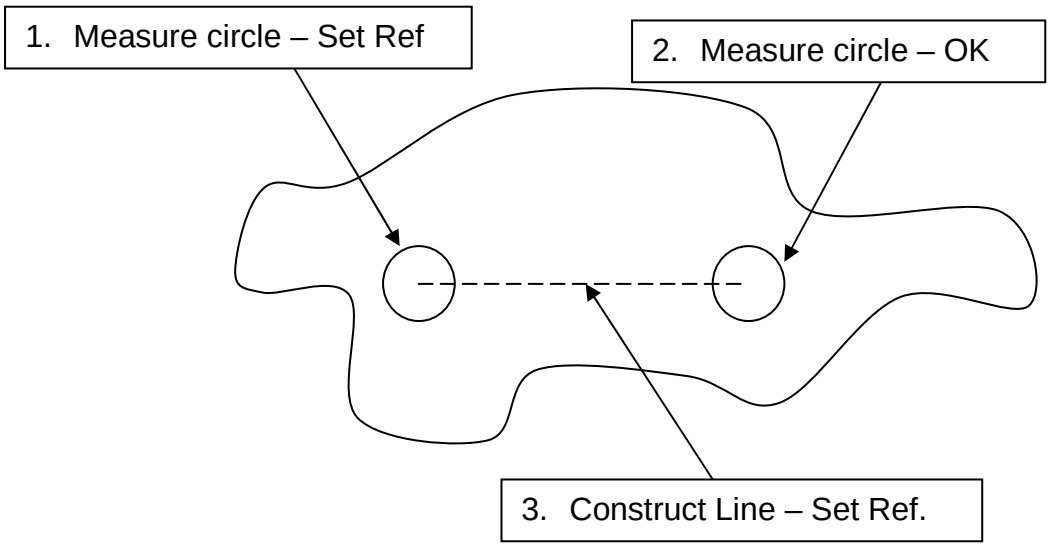


Fig. 6.0-13 A cylinder referenced to define the X axis of a component

The axis of the cylinder defines both the rotations about, and the translations along the Y and Z axes. This can be a useful technique for defining the orientation and position for turned components.

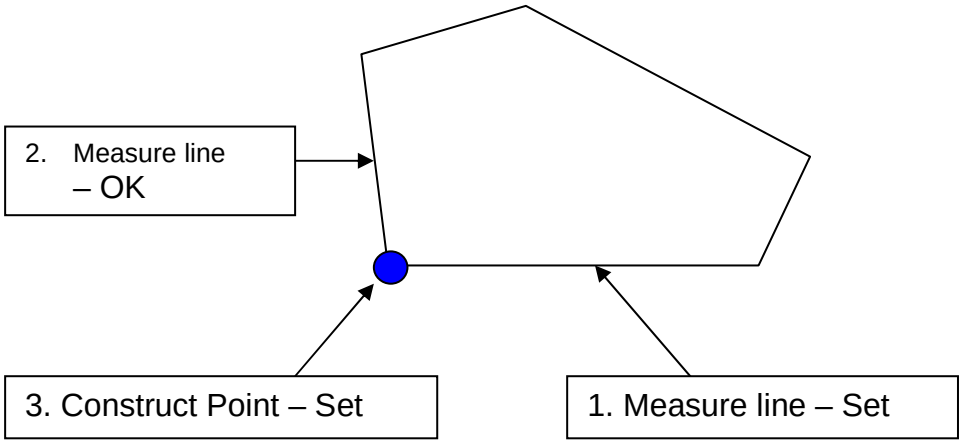
If the component being measured does not include these features.
There are other common features that can be used as references these are as follows:

1. No straight edge for alignment



www.EngineeringBooksPdf.com

2. No Suitable Hole



6.1 Basic Alignment

To measure a block type component like the Aberlink testpiece

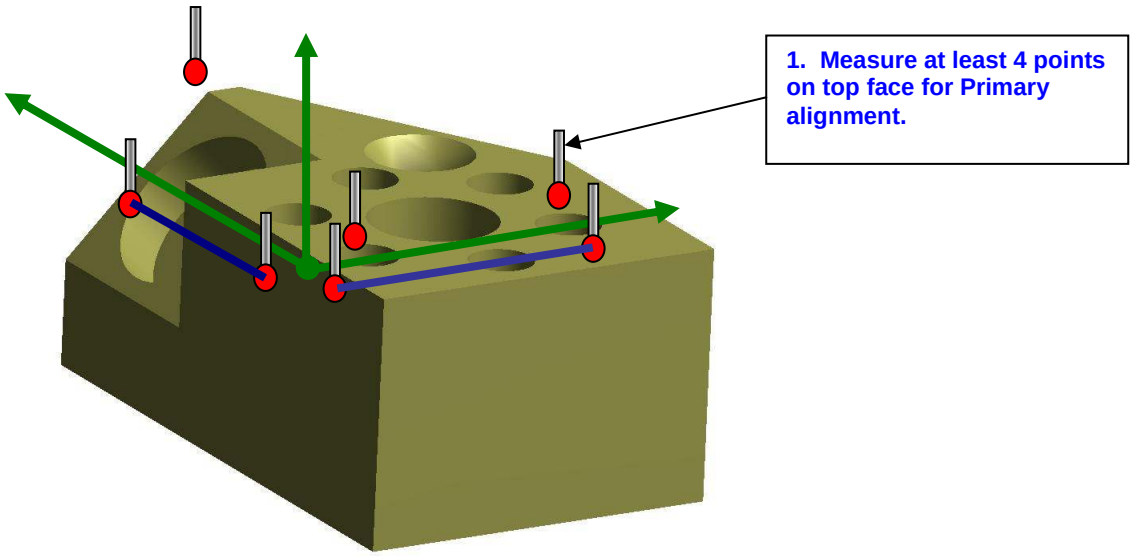


Fig. 5.1-1 Basic 3-2-1(2) Alignment

1. Measure a plane on the top of the block, by clicking on the measure ‘plane button’. When the measure plane window opens take 4 points on the top face (Plane 1). Press the ‘Set Ref’ button (this will align the Z axis 90° to the measured plane this is our **Primary alignment**) then click ‘OK’

2. Next measure a line along the side of the block, by clicking on the measure ‘line button’. When the measure line window opens take 3 points along the side of the part (Line 2). Press the ‘Set Ref’ button

www.EngineeringBooksPdf.com

(this will rotate the axis system to align the Y axis to the side of the part this our ***secondary alignment.***) then click 'OK'

3. The last thing we need to do is to measure a line along the back face of the part, by clicking on the measure 'line button'. When the measure line window opens take 3 points along the back face of the part (Line 3). Press the 'Set Ref' button.(this will create an ***origin*** (zero point) in the corner where the two lines intersect each other on the top face) then click 'OK'

www.EngineeringBooksPdf.com

6.2 Plane & two circles Alignment

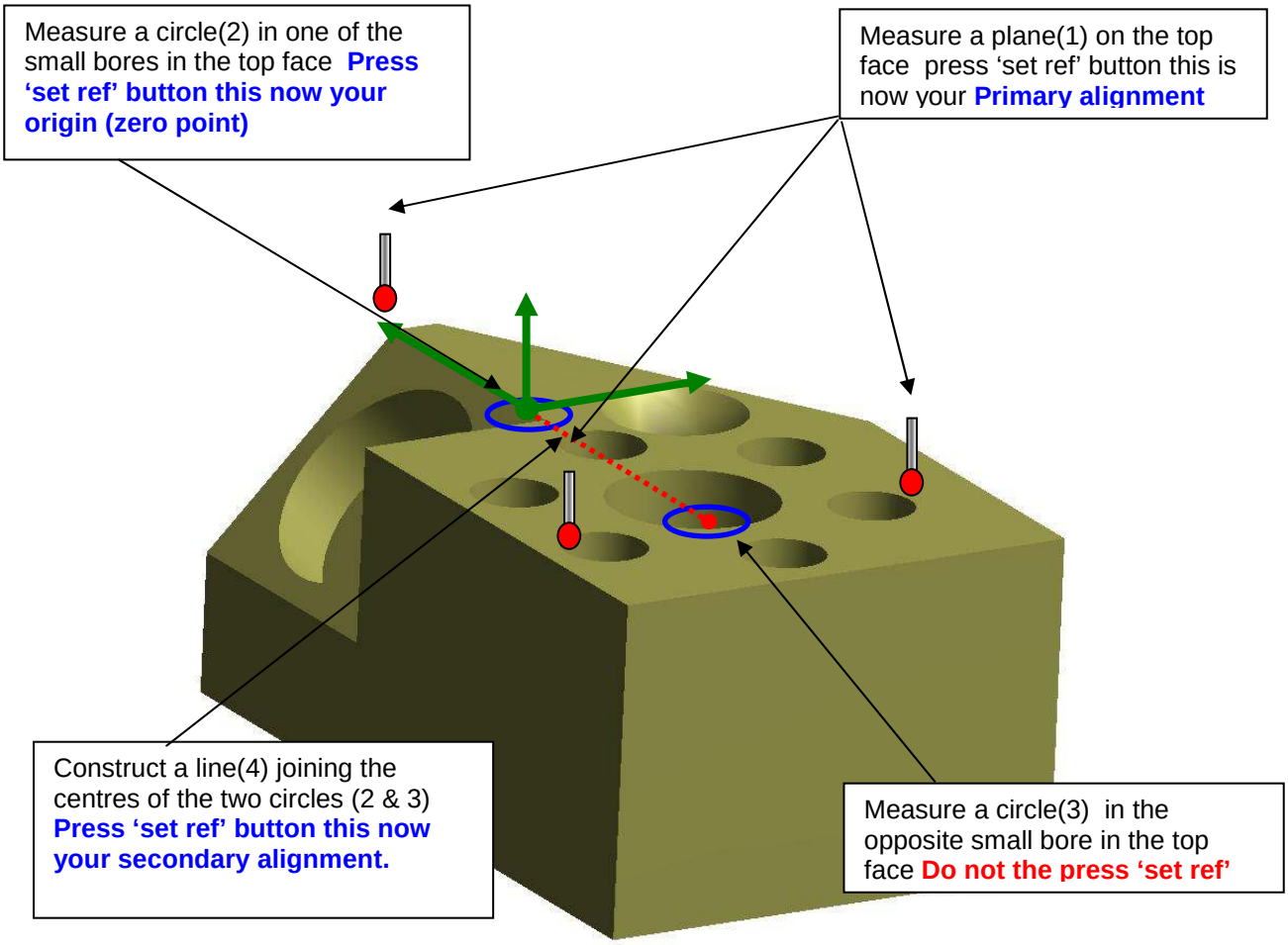


Fig. 6.2-1 Plane & two circles Alignment

1. Measure a plane on the top of the block, by Clicking on the measure 'plane button'. When the measure plane window opens take 4 points on the top face (Plane 1). Press the 'Set Ref' button (this will align the Z axis 90° to the measured plane, this is our ***Primary alignment***) then click 'OK'.

2. Measure a circle, one of the small bores, by Clicking on the measure 'circles button'. When the measure circle window opens take 4 points in one of the small bores (circle 1). Press the 'Set Ref' button this now is your ***origin (zero point).***

3. Measure the small bore opposite the one you just measured, by clicking on the measure 'circles button'. When the measure circle window opens take 4 points (circle 2). Do not press the 'Set Ref' button as we are going to use this circle to construct a line.

www.EngineeringBooksPdf.com

4. The last thing we are going to do is to align the Y axis of the CMM to a constructed line between the two circles. By clicking on the measure 'line button' when the measure line window opens , click on the 'construct button' then select circle 1 by left hand clicking on it in the graphic window. You will now see a window which asks you if you want to construct a line between two points . Click 'YES' select the other circle in the same way by left hand clicking on it in the graphic window. You now have a Constructed line, press the 'Set Ref, button and then click 'OK' this will align the Y axis the along constructed line this is our *secondary alignment*.

6.3 Aligning components using a cylinder

To measure a flange type component

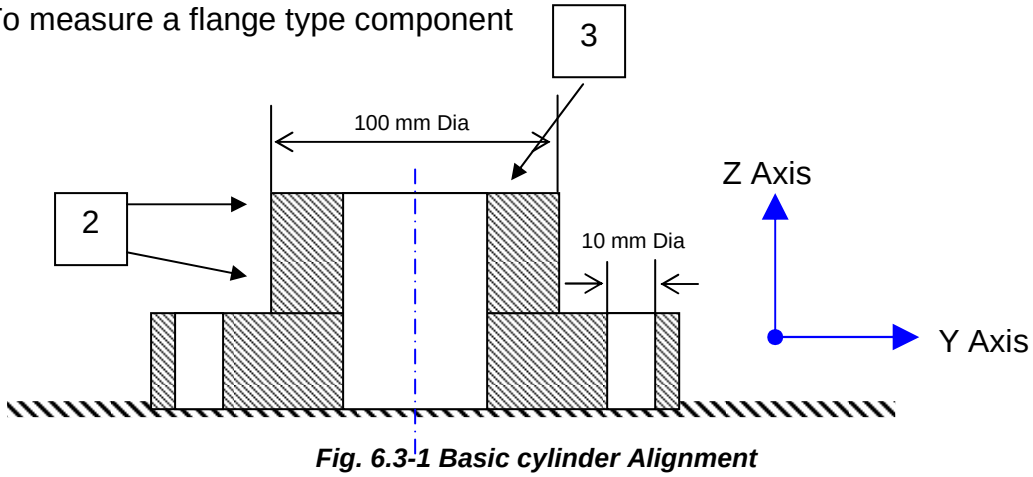


Fig. 6.3-1 Basic cylinder Alignment

1) Place the part on the table with the 100mm diameter facing up & two of the 10.0 mm holes aligned along the Y axis of the CMM.

2) Measure the 100 mm O/D as a cylinder by clicking on the cylinder measure button / icon with left hand mouse button (LHMB) When the cylinder measure window opens take 4 point at the top of the cylinder in a circle and 4 measure points around the bottom of the cylinder. Press the 'Set Ref, and then click 'OK' the Z axis is now aligned to the centre line of the cylinder.

3) Measure a plane on the end of the 100 mm cylinder, by clicking on the measure 'plane button'. When the measure plane window opens take 4 points on the top of the 100mm cylinder Do not press the 'Set Ref' button (this will be used as a working plane to project the point into) and click 'OK'.

www.EngineeringBooksPdf.com

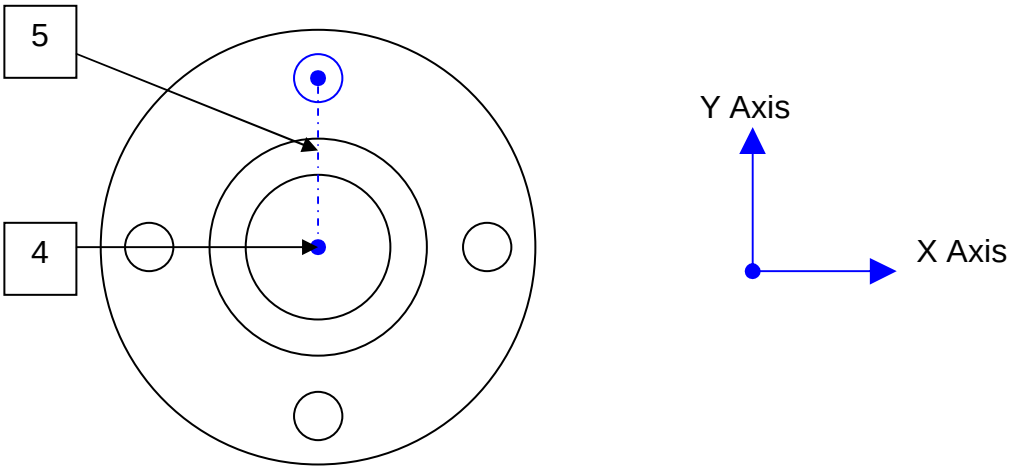


Fig. 6.3-2 Basic cylinder Alignment top view

4) Next we are going to construct a datum at the point where the cylinder meets the plane(X,Y datum centre of cylinder, Z datum on the top plane). Click on the measure 'point button' when the measure point window opens click on the 'construct button' then select the plane by left hand clicking on it in the graphic window. You will now see a window asking you if you want to construct a intersection point. click 'YES,' select the cylinder by left hand clicking it in the graphic window, you have now constructed a point, press the 'Set Ref', and then click 'OK' this zeros X,Y & Z axis.

5) The last thing we need to do is to align the Y axis through one of the 10mm holes. Click on the measure 'circle button'. When the measure circle window opens take 4 points in the 10mm hole at the 12 o'clock position then click 'OK'. You now have a 10mm circle in our working plane. We are now going to align the Y axis of the CMM via a constructed line between the datum point and the 10mm hole. Click on the measure 'line button'. When the measure line window opens, click on the 'construct button' then select the datum point by left hand clicking on it in the graphic window, you will now see a window which asks you if you want to construct a line between two points click 'YES.' Select the circle in the same way by left hand clicking on it in the graphic window. You now have a constructed line, press the 'Set Ref', button and then click 'OK' to align the Y axis along the constructed line.

The component is now aligned true to the centre line of the 100mm cylinder (Primary alignment) with the Y axis going through the 10mm hole (secondary alignment) with the X Y Z axis zeroed where the cylinder meets the top face.

www.EngineeringBooksPdf.com

6.4 Drive shaft (aligning along shaft's centre line)

Place the shaft on the CMM table with the shaft facing down the table (along the Y axis)

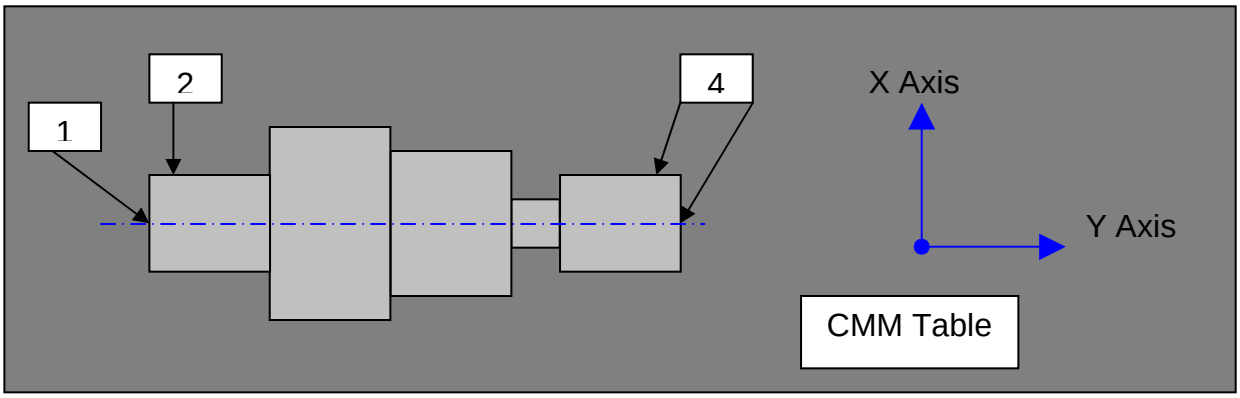


Fig. 6.3-1 Basic drive shaft Alignment

As the main datum is the centre line of the shaft we need to align the Y axis of the cmm to the centre line to do this:-

1) Click on the measure 'plane button' when the measure plane window opens take 4 points on one end of shaft Click 'OK' ,this is plane 1.

2) Click on the measure 'circle button'. When the measure circle window opens take 4 points around the end of shaft by plane 1, circle (2) is automatically projected into plane 1. Press the 'Set Ref, button this sets the XYZ datum on this end of the shaft (Zero point) click 'OK'.

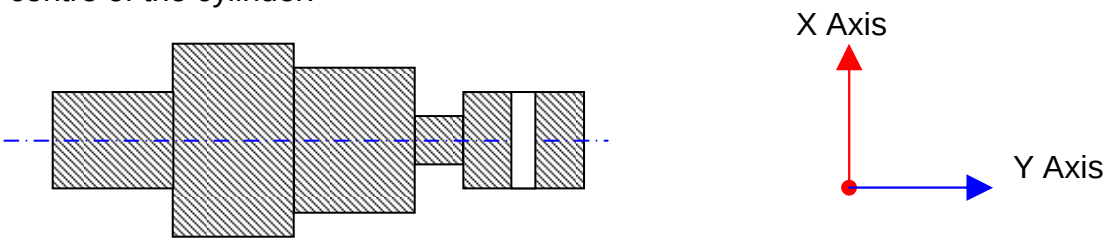
3) Repeat this process for the other end of the shaft but **do not press** the 'Set Ref', button this time you now have 1 plane and 1 circle at each end of the shaft, we can make a centre line by jointing these two circles together (constructed) and by pressing 'Set Ref' this is our (Primary alignment) .

4) To do this click on the measure 'line button' when the measure line window opens , click on the 'construct button' then select one of the circles by left hand clicking on it in the graphic window, you will now see a window which asks you if you want to construct a line between two points, click 'YES' select the other circle in the same way by left hand clicking on it in the graphic window. You now have a constructed line, Press the 'Set Ref, button and then click 'OK' this will align the Y axis along constructed line.

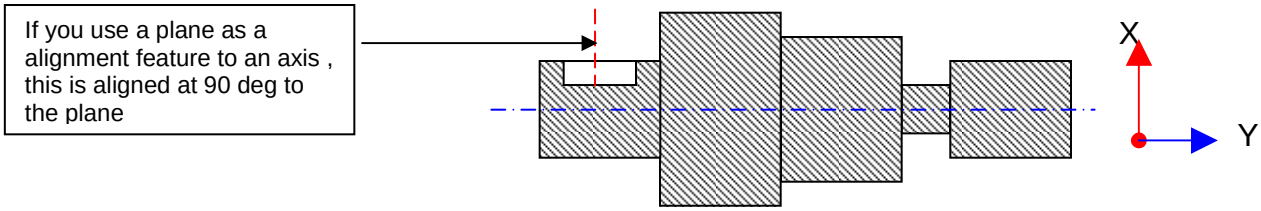
If the shaft has any cross holes, keyways or flats these can be aligned to one of the other axes (secondary alignment, Timing).

www.EngineeringBooksPdf.com

To do a secondary alignment using a cross hole measure the cross hole as a cylinder press the ‘Set Ref’ button, click ‘OK’ this will align either Z or X axis to the centre of the cylinder.



To do a secondary alignment using a keyway or flat measure the bottom of the keyway or flat as plane press the ‘Set Ref’ button click ‘OK’ this will align either Z or X axis to a line 90 deg to the plane.



The component is now aligned true to the centre line of the shaft (Primary alignment) with the Z or X axis going through the cross hole or at 90 deg to the flats (secondary alignment) with the X Y Z axis zeroed in the centre of the shaft at the front end.

www.EngineeringBooksPdf.com

6.5 Optimising Alignment

6.5.1 Optimising to Horizontal and Vertical Dimensions

It is possible with some shapes that the optimum alignment for the component is not achieved by referencing any combination of measured features. It is possible to select certain horizontal and vertical dimensions on the measured component, and then optimise the alignment of the component in order to minimise the errors of these dimensions.

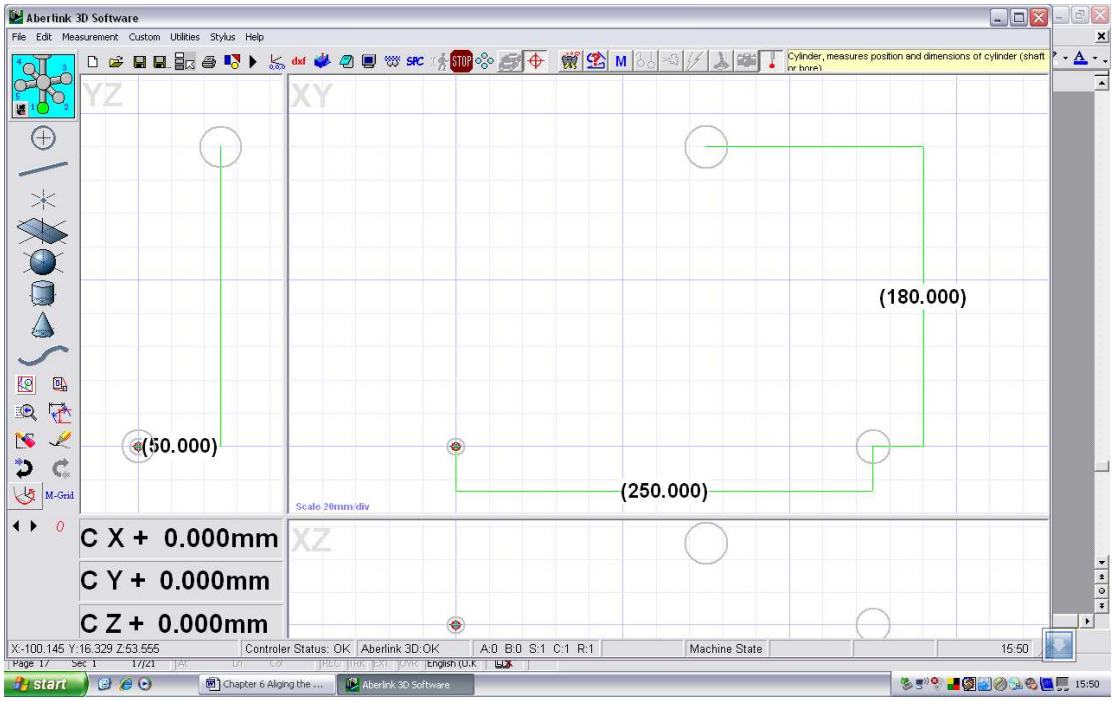


Fig. 6.5-1 Align part to 250 , 180 , 50 Dimensions

Firstly select the horizontal and vertical dimensions that you wish to use to optimise the alignment of the component, by right mouse clicking on the dimension to bring up the dimension detail window.

www.EngineeringBooksPdf.com

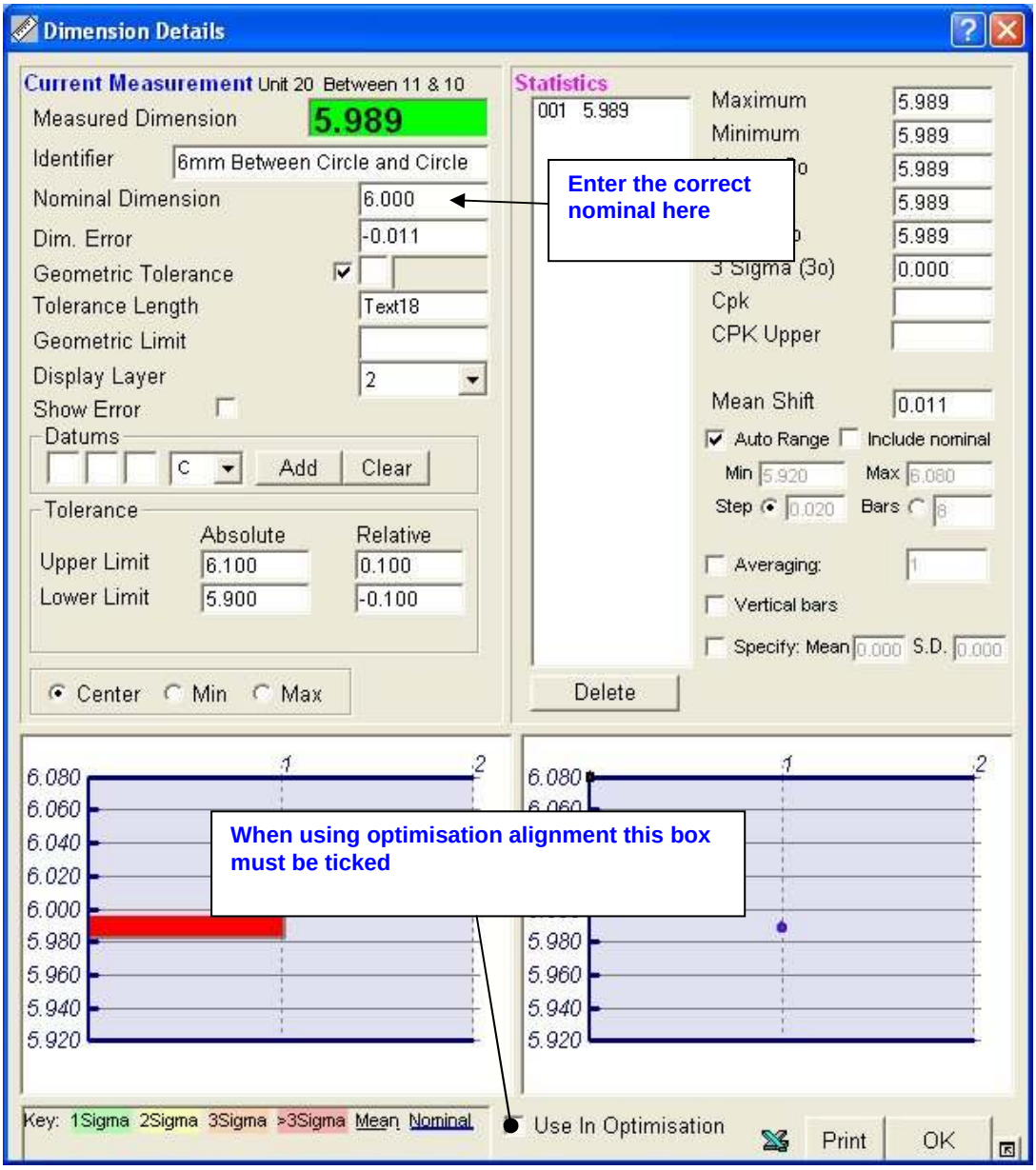


Fig. 6.5-2 Dimension Details

Type in the correct value for the nominal dimension, if necessary, and then tick the box labelled ‘ Use in Optimisation’. Click on OK to return to the main screen.

When you have selected all the dimensions that you wish to use in the optimisation, click on the Grid Shift button.

Fig. 6.5-3 Grid Shift

www.EngineeringBooksPdf.com

When aligning to horizontal and vertical dimensions it is possible to select either ‘Nominal’ or ‘Tolerance’.

‘Nominal’ will align the part to get all of the selected dimensions as close as possible to their nominal value.

‘Tolerance’ will align the part to try and get all of the selected dimensions as close as possible to their tolerance bands.

Then click on ‘Start’. The software will take a few seconds to calculate the best alignment, and when it has finished click on OK.

Note:- That the minimum number of dimensions should be selected that best define the rotation required for the component, ie. Typically this will be one dimension for X, one for Y. This will help to minimise the calculation time required, although this is primarily dependant on the number of features that have been measured.

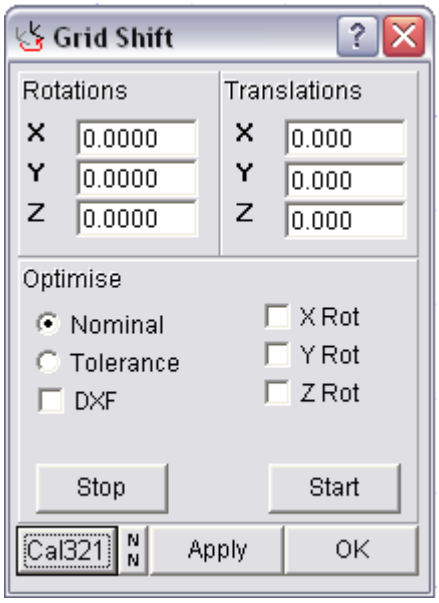


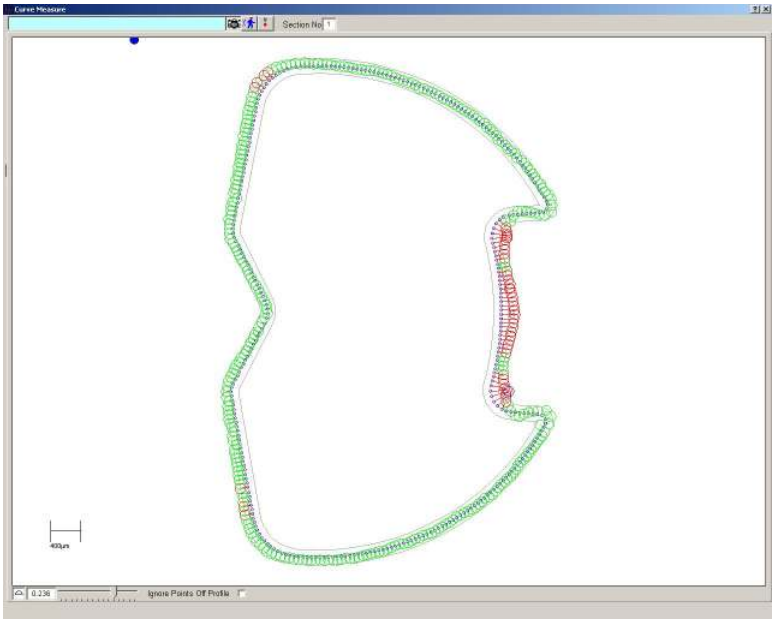
Fig. 6.5-4 Grid Shift Window

6.6 Optimising to a DXF File

If the component shape has been measured as a Curve profile, it is possible to import the DXF file for the shape, and perform a best-fit optimisation to get the measured points to fit the DXF file.

The software will report the error of each measured point from the corresponding closest point on the DXF file.

www.EngineeringBooksPdf.com

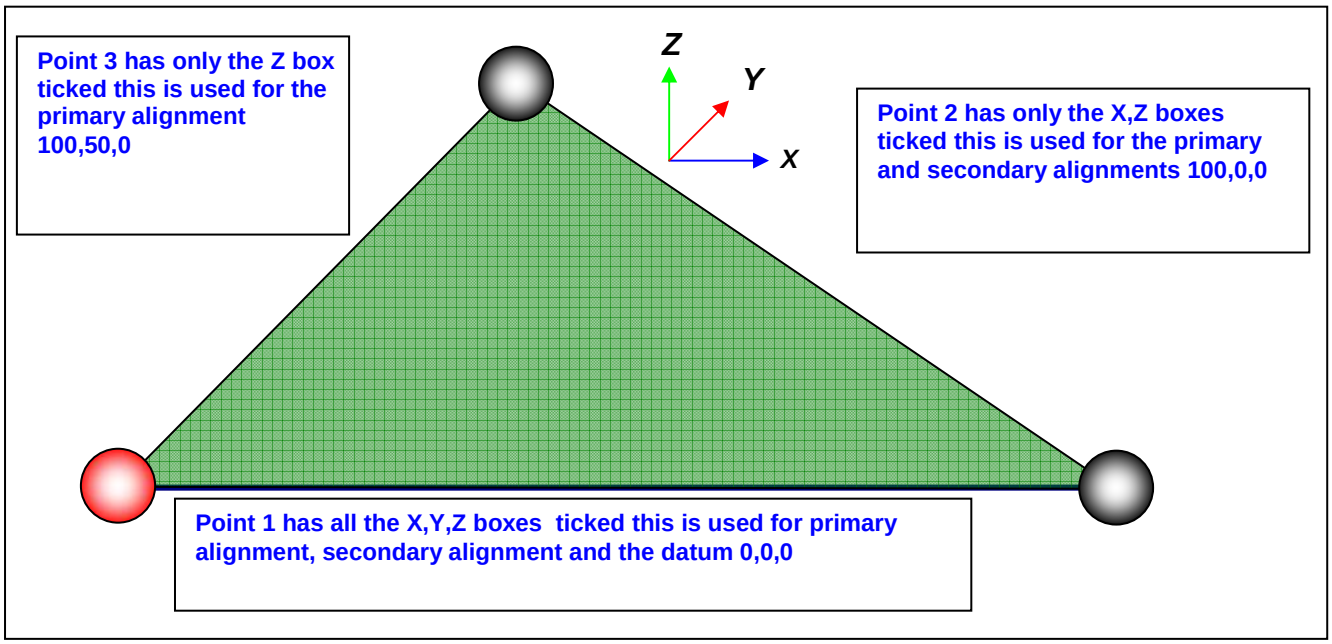


This is discussed fully in chapter (43.0 INPUTTING AND OUTPUTTING DXF FILES.

6.7 3,2,1 Optimised Alignment

You can align using three points in space using 3,2,1 optimised alignment.

- All of the three points are used for the primary alignment of the part (plane).
- Two of the three points are used for the secondary alignment / timing of the part (line).
- One of the three points is used to set the datum on the part (point).



If you have three feature on the part you are measuring which have known X,Y,Z position in space, like a sphere or a circle measure within a plane.

www.EngineeringBooksPdf.com

It is possible to align the part using the 3,2,1 optimised alignment.

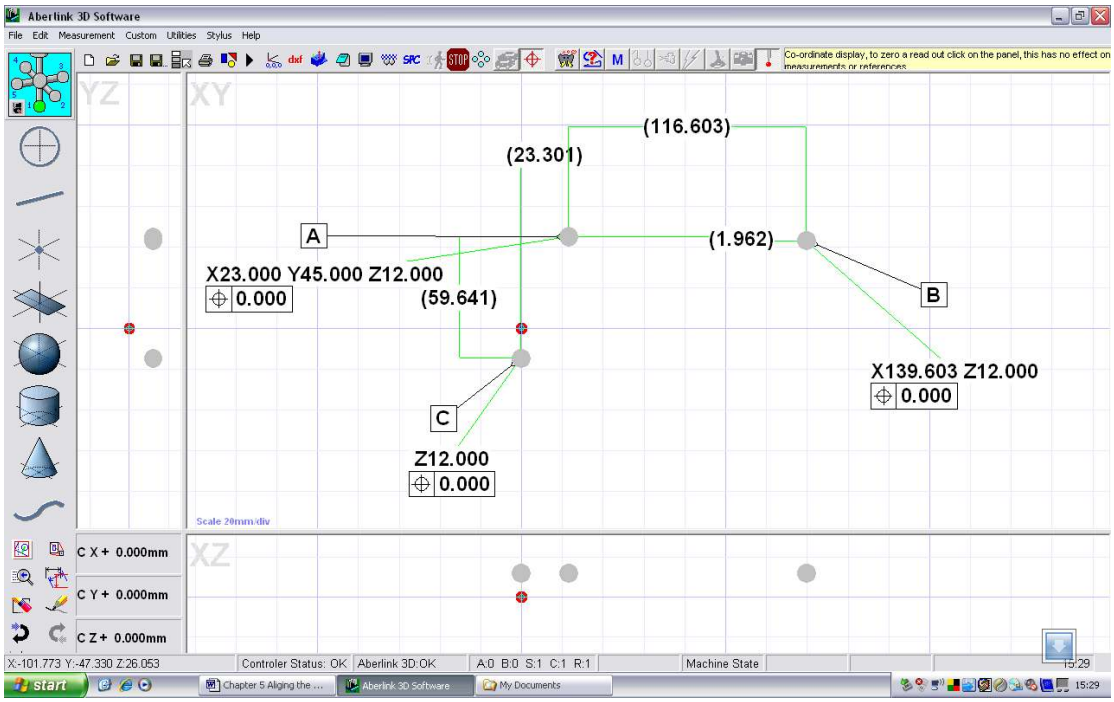


Fig. 6.7-1 3,2,1 optimised alignment

The position shown on the drawing for A is X=0, Y=0, Z=0 for B is X=100, Z=0 for of C is Z=0

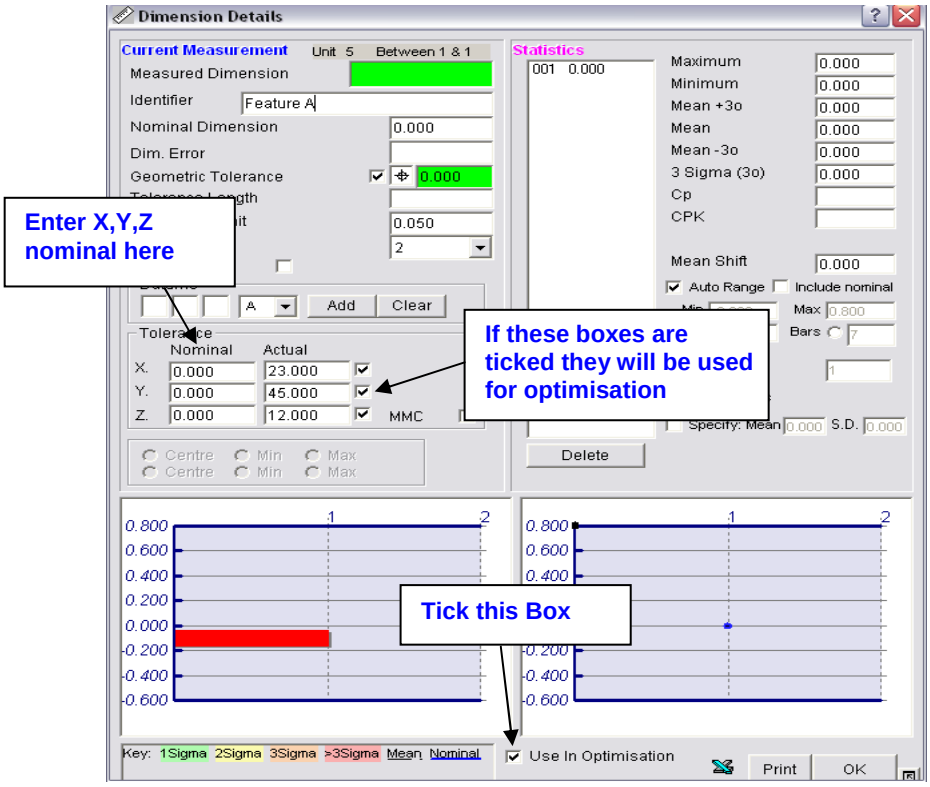


Fig. 6.7-2 3,2,1 Tick boxes for optimised alignment

www.EngineeringBooksPdf.com

1) Measure the true position of features A,B,C.
2) Display the position on the screen.
3) In the Dimension details window enter the correct nominal in the X,Y,Z boxes.
4) At the bottom of the window tick the ‘use in optmisation’.
5) There is a tick box next to X,Y.Z nominal boxes. For feature A tick all the boxes, for feature B Tick the X & Z boxes and for feature C only tick the Z box.

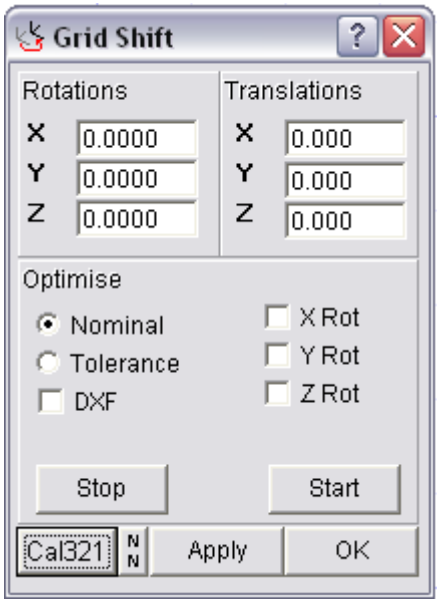
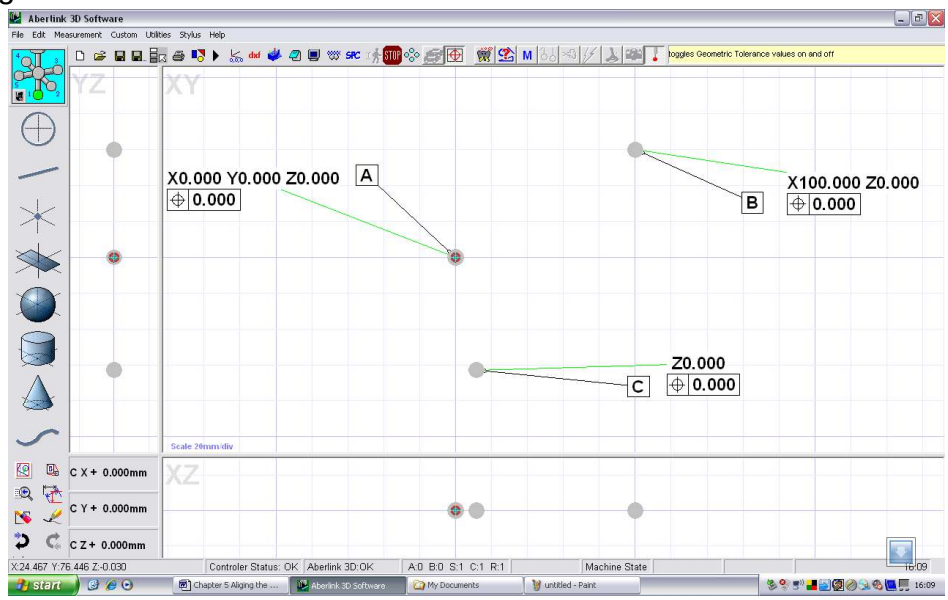


Fig. 6.7-3 Grid Shift Window

6) Click on the ‘Grid Shift’ icon
7) In the ‘Grid shift’ window click on the ‘cal321 button’ the software will now align the part using the ticked nominal.
8) There are always four solutions for each alignment by clicking on the ‘NN’ button next to the ‘cal321 button’ the software to cycle through each alignment 1 to 4.



www.EngineeringBooksPdf.com

Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

www.EngineeringBooksPdf.com

7.0 CREATING A CNC PROGRAM – a general introduction

When using the Aberlink 3D CNC software there are 2 ways to create a CNC program, as follows:

7.1 Teach and Repeat

By driving the machine around using the joystick and measuring a component you are also creating a program that can afterwards be run automatically. How to use the joystick is detailed in section 4.

7.2 Off-Line Programming

Sometimes it will be quicker simply to type in the size and position of a feature to be measured rather than to measure it directly. Alternatively it may be that you simply wish to create a program without using the CMM. In this situation you can enter the feature details using the keyboard and click on 'Template from Details given'. This is described in detail in each of the Measurement Unit sections.

Programs can even be created by a combination of the two methods, and often this can be the most efficient solution.

If you wish to run the program automatically you will need to define the position and alignment of the component by measuring certain features and setting them as references. This idea is discussed the last chapter.

When you are ready to run the program read chapter 27.0 THE PLAY FUNCTION – running a program automatically

www.EngineeringBooksPdf.com

8.0 THE PLANE UNIT

The plane unit is used to measure flat surfaces, such as one face of a cube, by taking 3 or more measurement points spread across the surface.

Planes can also be constructed through 3 or more previously measured points.

You can also construct a plane, offset from another plane by a fixed distance, which can be entered via the keyboard.

8.1 Measuring Planes Directly

Background
Planes are used to measure flat surfaces on components, and the software will construct the 'best fit' plane through the measurement points taken.

In order to apply the stylus compensation in the correct direction, the software will look at the direction of motion of the probe when taking the measurement points.

If you measure a plane and set it as a reference, this will align the component to this plane, ie. You will have defined 3 of the 6 degrees of freedom for the component (see 6.0 ALIGNING THE AXES OF A COMPONENT). This allows features that are perpendicular to the plane to be measured as 2 dimensional lines and circles.

8.2 Teach and Repeat

Method
To measure a plane, click on the Plane Measure button, from the main screen:

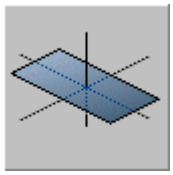


Fig. 8.1-1 The Plane Measure Button

www.EngineeringBooksPdf.com

This will bring up the Plane Measure window:

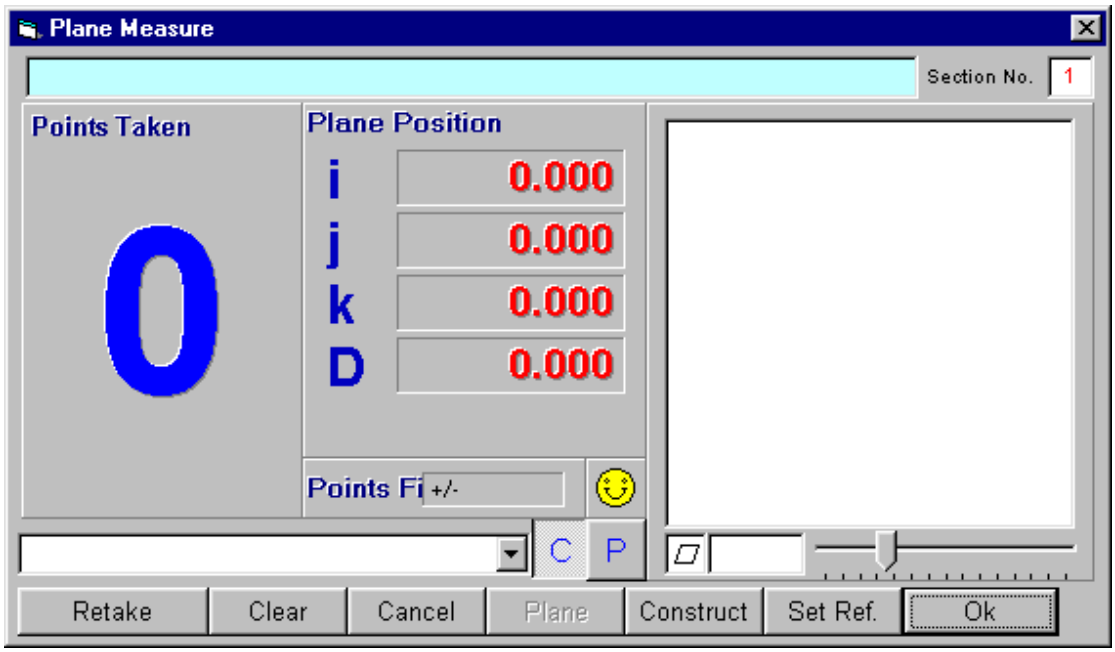


Fig. 8.1-2 The Plane Measure Window

Now take measurement points on the surface that you wish to measure. After 3 points have been taken a pictorial representation of the plane will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The I,J and K values displayed represent the direction vector of the nominal to the plane and the D value represents the distance between the plane and the current origin. (**Normally this information will not be particularly relevant**).

www.EngineeringBooksPdf.com

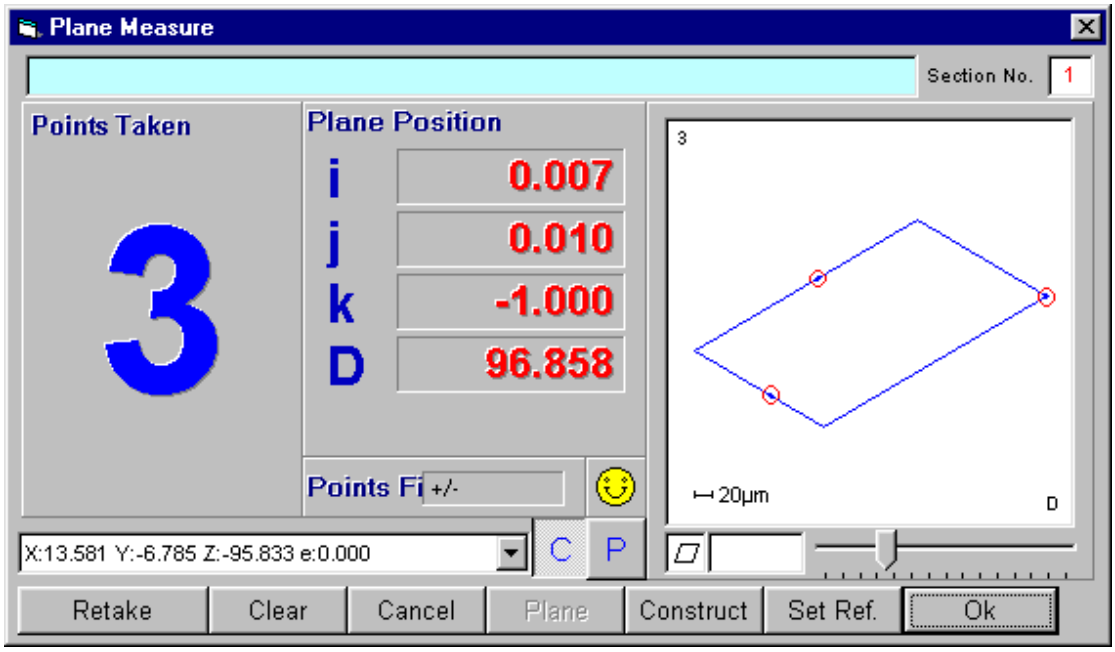


Fig. 8.1-3 A Graphical Representation of a plane after 3 measurement points

A plane can always be calculated to fit through three points, therefore the first 3 points will always produce single bleeps and the yellow face will remain smiling. Also the 'Points Fit' box next to the yellow face will remain blank and the flatness value for the plane, shown under the graphical representation, will show zero.

It is therefore good practise to take at least one more point. On the fourth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated 'best fit' plane and a value for the geometric flatness of this plane. Also there is now some meaning to the 'bad point' sound and face warnings. If the last point taken doesn't fit the plane very well, then the face will start to frown. If it doesn't fit at all well, the face will become sad, and the computer will make the 'bad point' sound. The threshold values for the smile and frown warnings can be user specified, see 28.1.5 Smile / Frown Threshold

In general the more points taken on a plane, the more information can be gleaned, and a minimum of 4 points is recommended. Similarly the points taken should cover as much of the surface of the plane as possible.

The graphical representation of the plane shows the points taken as a red circle with a blue isometric rectangle representing the 'best fit' plane through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

www.EngineeringBooksPdf.com

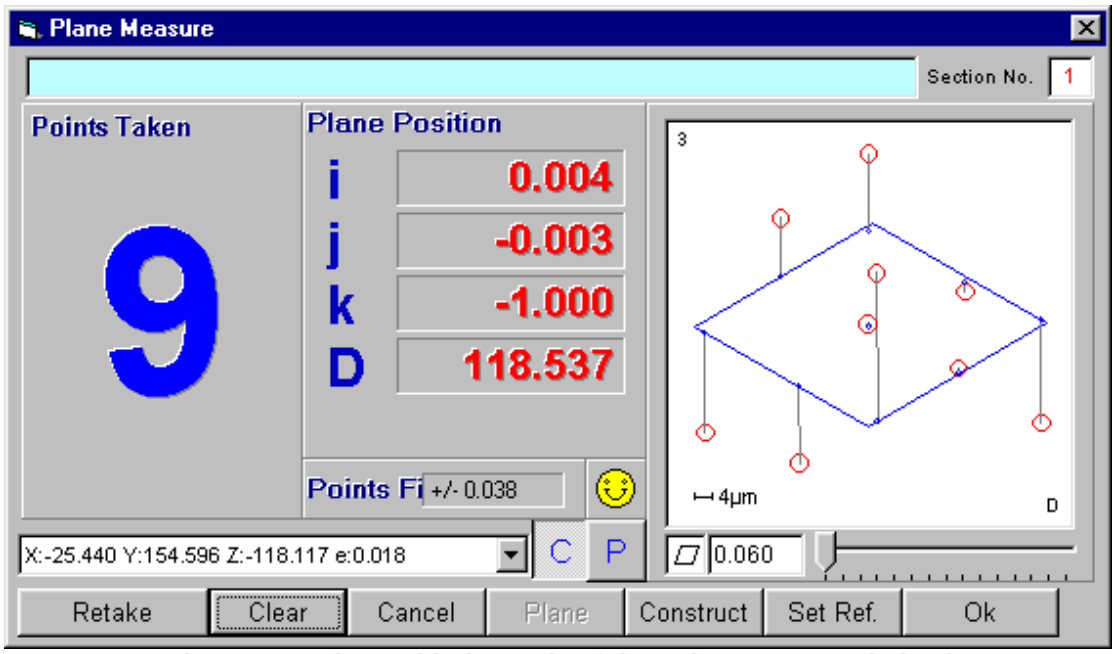


Fig. 8.1-4 A Plane with the scale of the point errors maximised

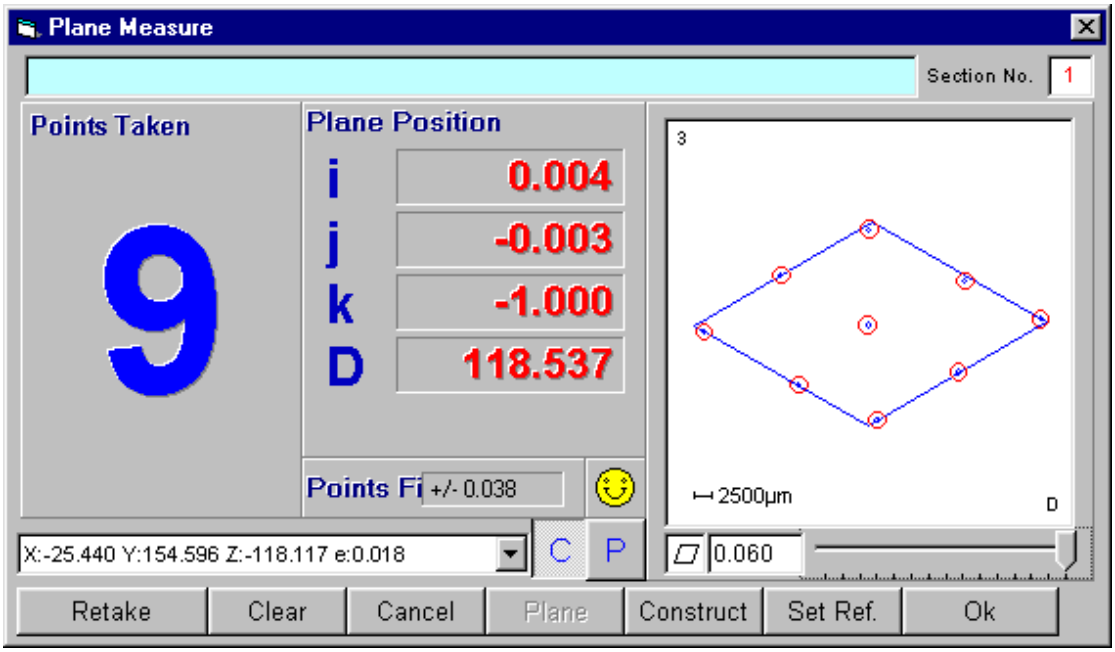


Fig. 8.1-5 A Plane with the scale of the point errors minimised

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See 16.0 ERASING DATA POINT.

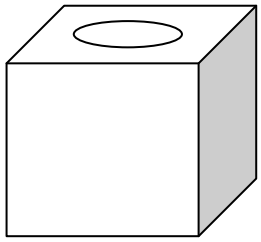
8.2. Off-Line Programming

Instead of directly measuring the plane using the probe it is possible to enter the co-ordinates for the plane and let the software calculate a grid of points to be taken uniformly across the plane. In this situation note that the plane is always rectangular by default and that the points automatically calculated will

www.EngineeringBooksPdf.com

form a regular grid, even if there are other features or obstacles in the way:

For instance we want to produce an off-line program for the plane on the top face of a cube with a hole in it. Assume the cube size is 40mm and the hole diameter is 25mm. The circle has been set as a reference, so its centre is X=0, Y=0, Z=0.



Click on the Plane Measure button, from the main screen to bring up the Plane Measure window. Now instead of taking points with the probe we are going to type in the template details directly:

www.EngineeringBooksPdf.com

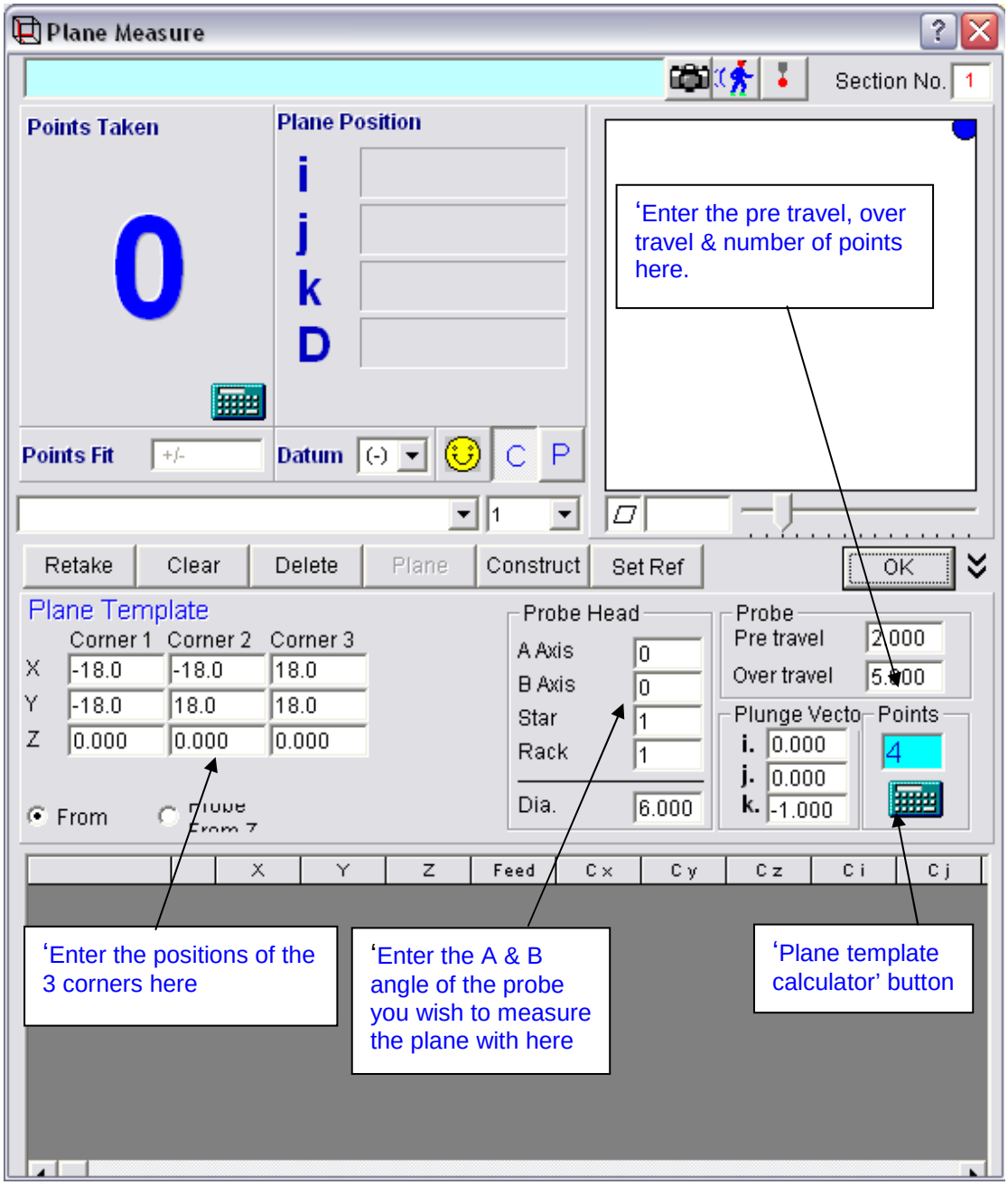


Fig. 8.2.2-1 Plane template window

A plane is defined by entering the positions of three of the four corners of the rectangle. In this case the front left corner of the plane is at X=-20, Y=-20, Z=0 but if we probed at this position we would catch the edge of the block. Therefore it may be wise to move in by 2mm hence we can enter the position for corner 1 as X=-18, Y=-18, Z=0. Similarly corner 2 could be X=-18, Y=+18 Z=0, and corner 3 X=+18, Y=+18, Z=0.

The pre-travel and the over travel will be set at their default values, but can be modified as required. Similarly you can change the probe head position and plunge vector (direction from which the probe will approach the plane) if required. The default number of points on the plane has been set to 4 so lets leave it at 4 for the moment.

www.EngineeringBooksPdf.com

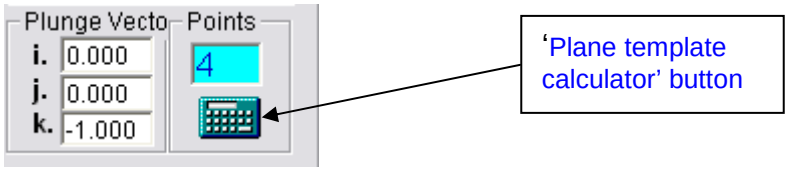


Fig. 8.2.2-2 Plane template calculator button

Now click on the 'Plane template calculator' button. The probe moves grid will now be automatically filled in, and the 4 points will be positioned at the corners of the plane defined:

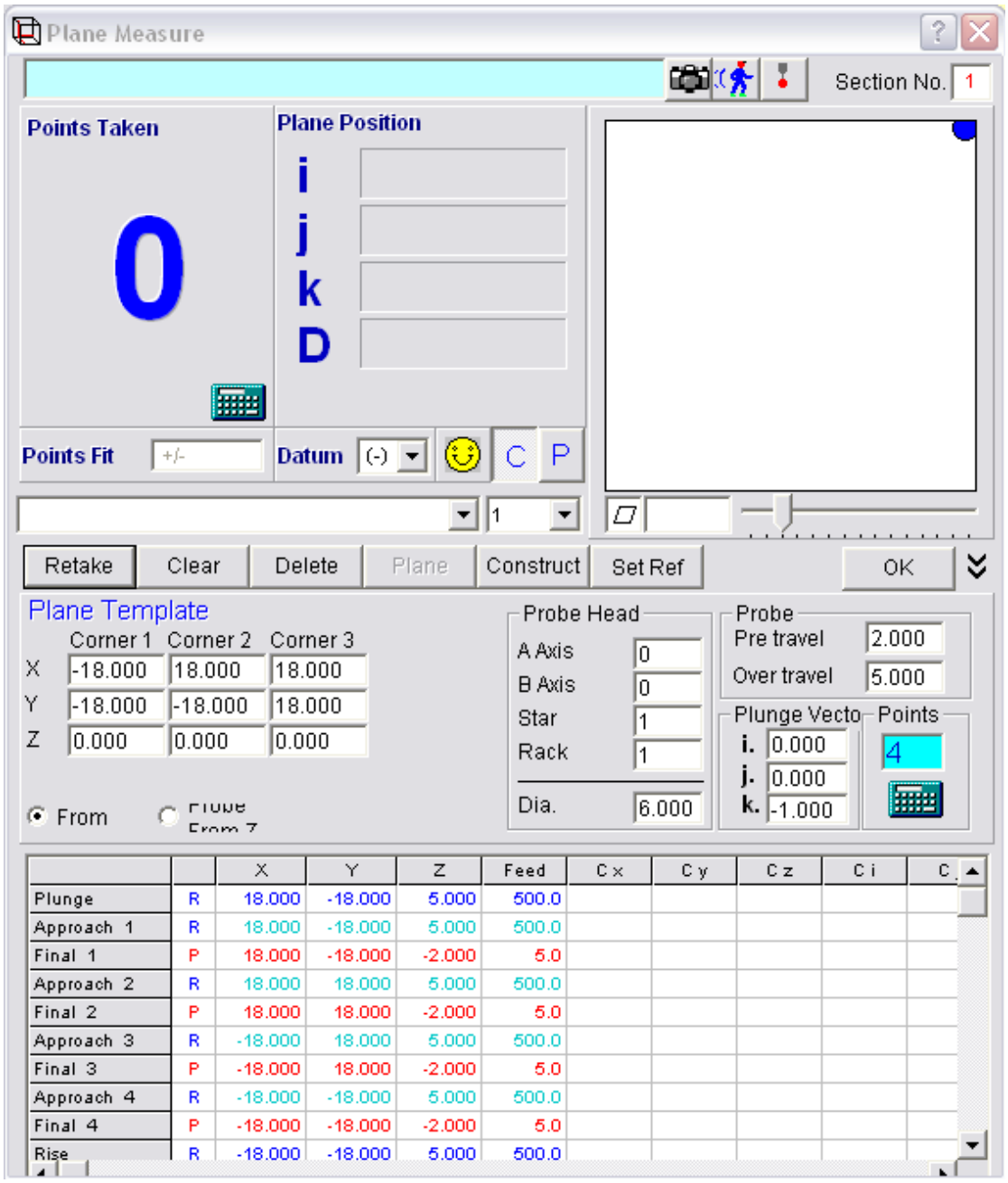
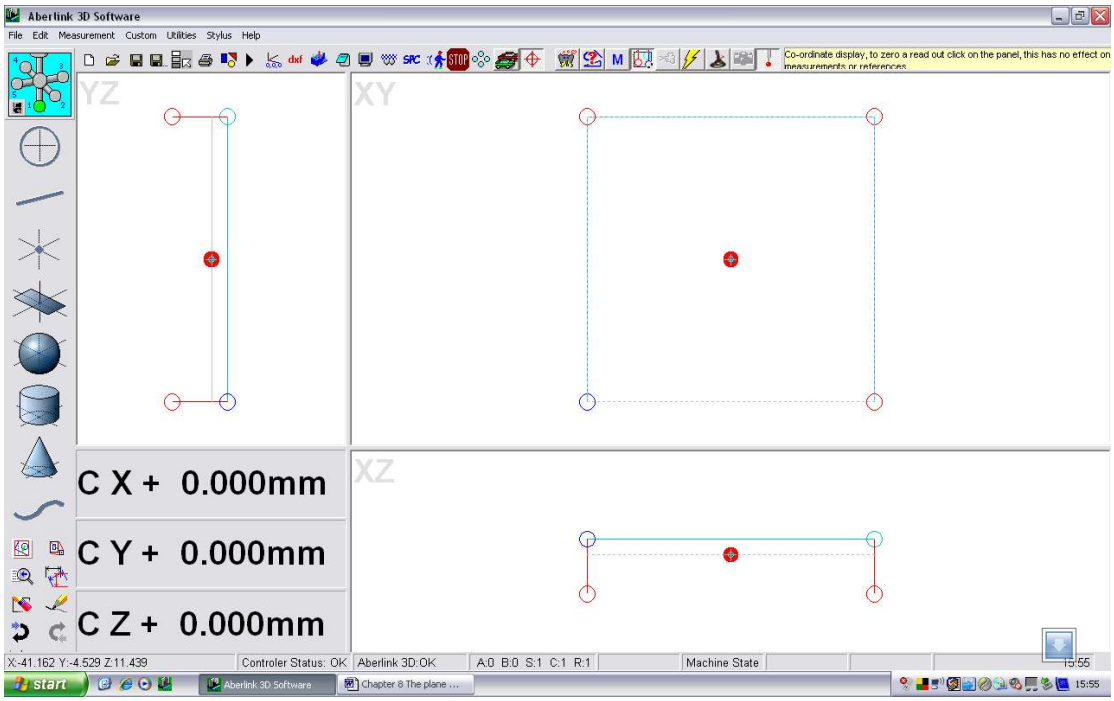


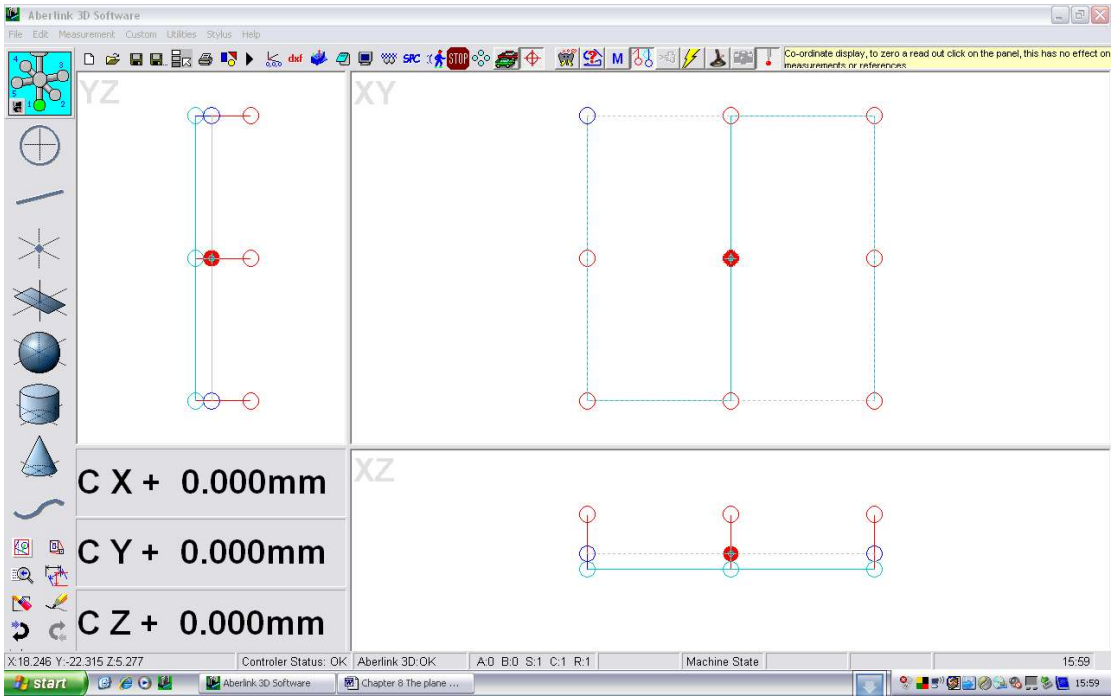
Fig. 8.1.2-3 Plane window showing the probe move grid

www.EngineeringBooksPdf.com

You can now click on ‘OK’. The plane will appear as a grey template in the main screen. If you click on the ‘Display Probe Contact Points’ button the measurement points will be displayed as dots:



Say that we now changed our mind and decided that measurement of the plane was far more critical than at first thought, and we now wanted to take 9 points. Recall the Plane Measurement window (by right clicking on the representation of the plane) and change the number of points to 9. Now click on the ‘Plane template calculator’ button again and the probe moves grid will be recalculated now with 9 points in it. Click on OK to return to the main screen:

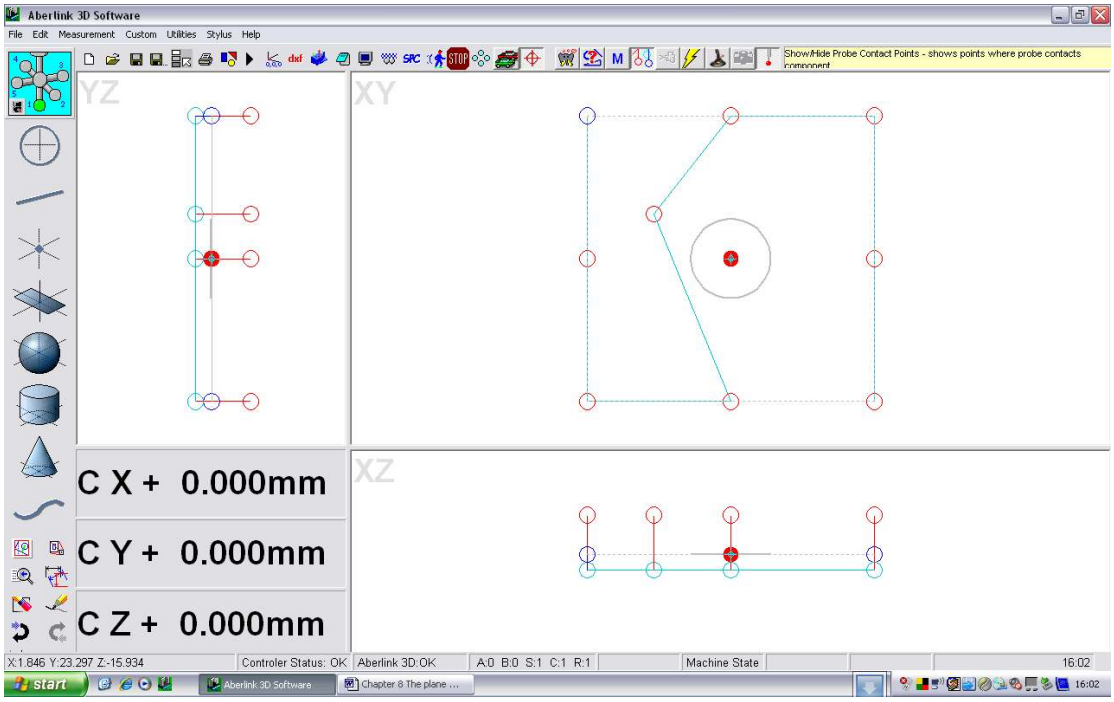


www.EngineeringBooksPdf.com

Note that the software has now produced a grid of 9 points across the plane, and that the central point will in fact try to probe down the hole in the cube.

There are 2 courses of action available:

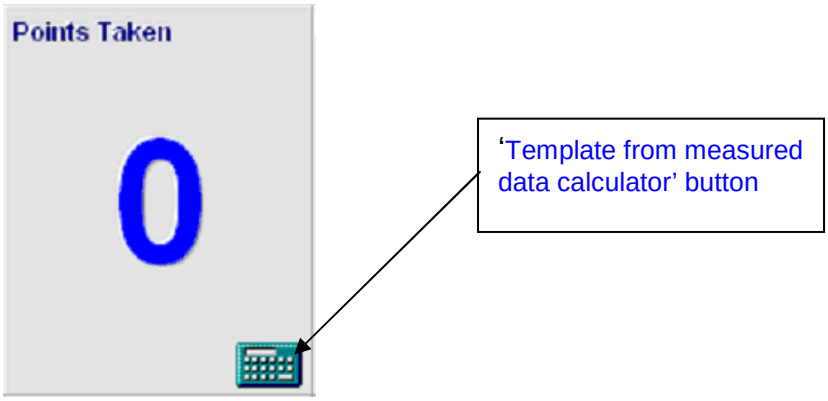
1. Move the point so that it doesn't probe down the hole.
Simply click on the red dot and drag it to a convenient point where it misses the hole. When you release the mouse button the point will be taken at that position:



2. Erase the point.
In this case this point is point number 5. Recall the Plane Measure window and in the probe moves grid left click on 'Approach 5', then right click and select 'Delete Cells'. Then repeat this exercise for the move called 'Final 5'.

8. 3 The 'Template from Measured Data calculator' button

www.EngineeringBooksPdf.com



When you create a program unit using the 'teach and repeat' method the program template is automatically calculated from measurement points taken.

If however, you need to re-calculate the template at any time, but cannot use the template details (for instance the points cannot be equi-spaced) then it is possible as follows:

- Clear the existing points, by clicking on the 'Clear' button.
- Take some new measurement points with the probe.
- Click on the 'Template from Measured Data' button to create a new program template from the points taken.

Note when you click on the 'Template from Measured Data calculator' button the probe moves grid is re-calculated using the points taken. These points are shown in the top left hand corner of the window.

It is bad practice to re-calculate the template for a feature that has been set as a reference once the program has been run.

8.4 When to OK a plane, and when to reference it.

Having measured a plane, you can either click on the 'OK' button or the 'Set Ref ' button.

If you OK the plane, the measurement window will close, and the program will return to the main screen.

If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position. See 17.0 MANIPULATING THE GRAPHICS

The graphical representation of the plane should be shown as a dashed rectangle in all of the 3 views. This representation will be pictured at whatever angle the plane is positioned relative to the machine table.

www.EngineeringBooksPdf.com

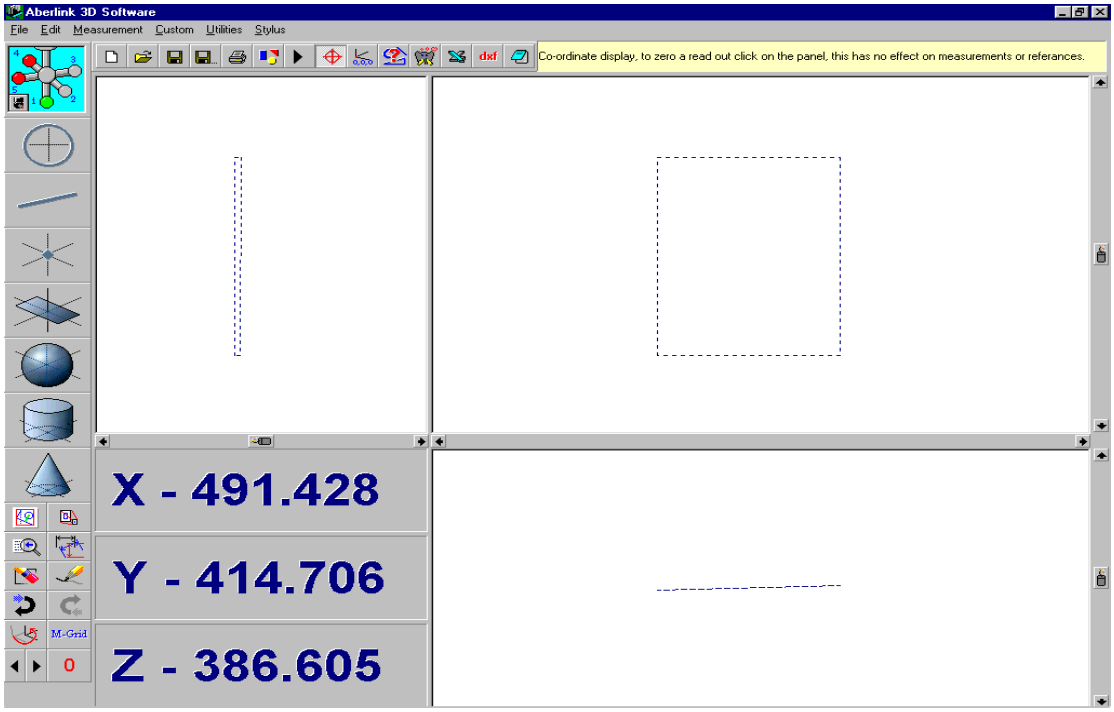


Fig. 8.4-1 The Main Screen showing a Plane which has been OK'd

If you set the plane as a reference plane by clicking on 'Set Ref' button if you then clicking on the 'OK' button the measurement window will close, and the program will again return to the main screen.
The graphical representation of the plane will again be shown as a dashed rectangle, but this time it will be aligned to the plane.

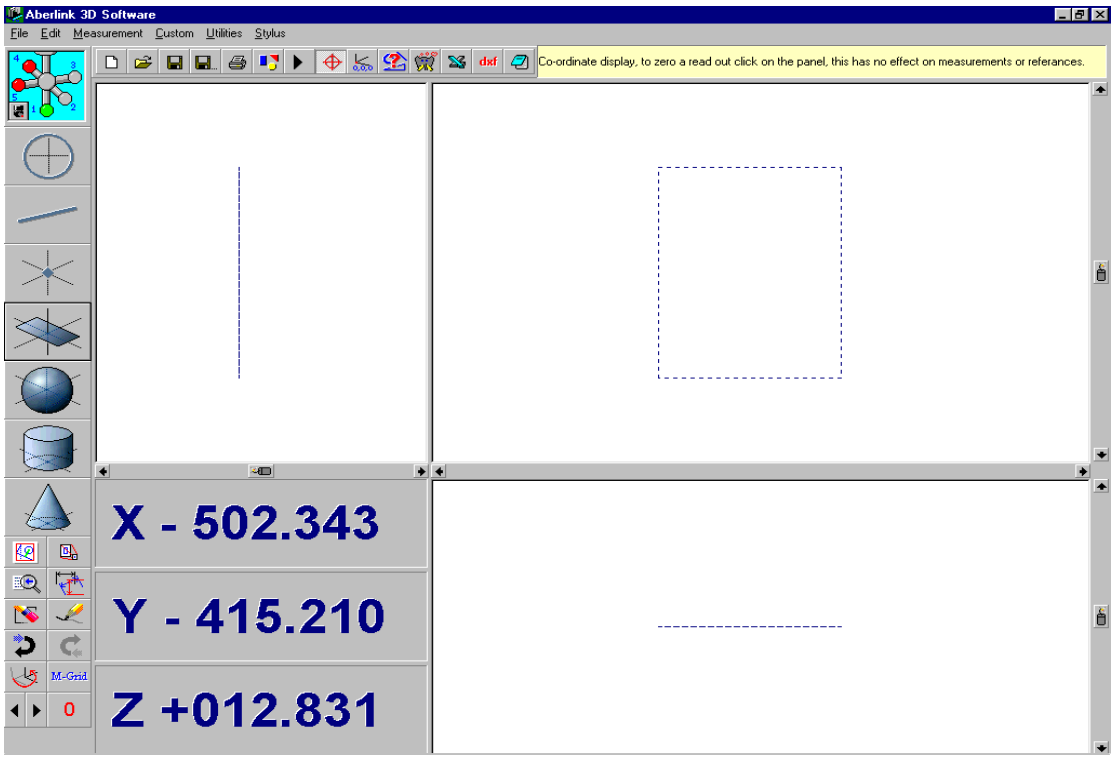


Fig. 8.4-2 The Main Screen showing a Plane which has been Referenced

www.EngineeringBooksPdf.com

The main reason for setting a plane as a reference is to define the alignment of the component in order for features, which are perpendicular to the plane, to be measured as 2 dimensional shapes (ie. lines and circles). Hence, if the top face of a cube is measured as a plane and set as a reference, then each of the side faces may be measured as lines, with all of the measurement points taken (at whatever depth) projected on to the reference plane to produce a straight line. Similarly, points taken in a hole perpendicular to the reference face may be projected on to the reference face to produce a 2 dimensional circle.

Lines and circles are a very quick and accurate method of inspecting a component. See chapters , The Line and Circle Units.

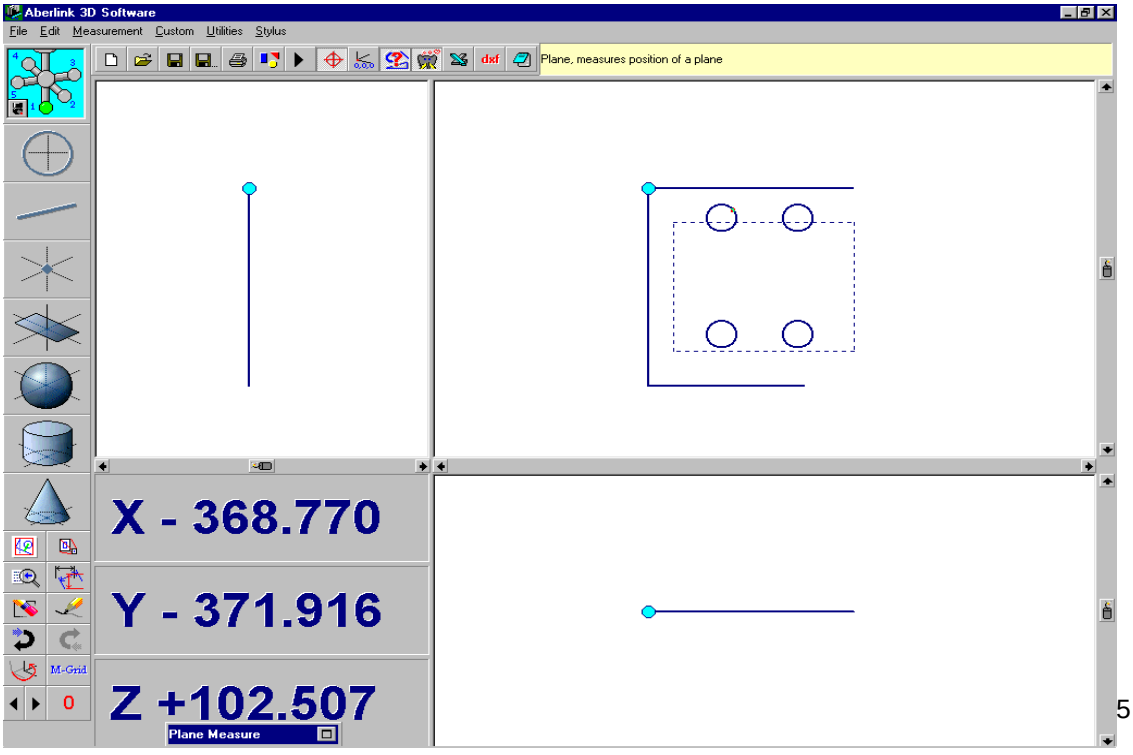
> ***Note*** *– Only one plane can ever be set as a reference at any point in time. If a second plane is set as a reference, this will override the initial reference plane, and the graphical representation will now re-align itself to the second plane.*

8.5 Constructing a Plane through previously measured points

As well as measuring planes directly, they can also be constructed through features which have previously been measured, or constructed themselves. For this purpose, circles and spheres may be considered as points, and lines, cones and cylinders can be considered as 2 points at the ends of their axis. A minimum of three points is required to construct a plane.

To construct a plane, click on the Plane Measure button, from the main screen. This will bring up the Plane Measure window. Now click on the ‘Construct’ button. The Plane measure window will now shrink to a small box at the bottom of the screen:

Fig. 8.5-1 The Main Screen showing a Shrunken Plane Measure Window



www.EngineeringBooksPdf.com

Now simply select the features through which you wish to construct the plane by clicking on them. After you have selected the first feature a prompt will appear to confirm what is happening:

Fig. 8.5-2 'Constructing a Plane' prompt

Click on OK, and then continue to click on the remaining features. When you have selected all the desired features bring the Plane Measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the plane will appear with values for the fit and flatness given if more than 3 points were selected.

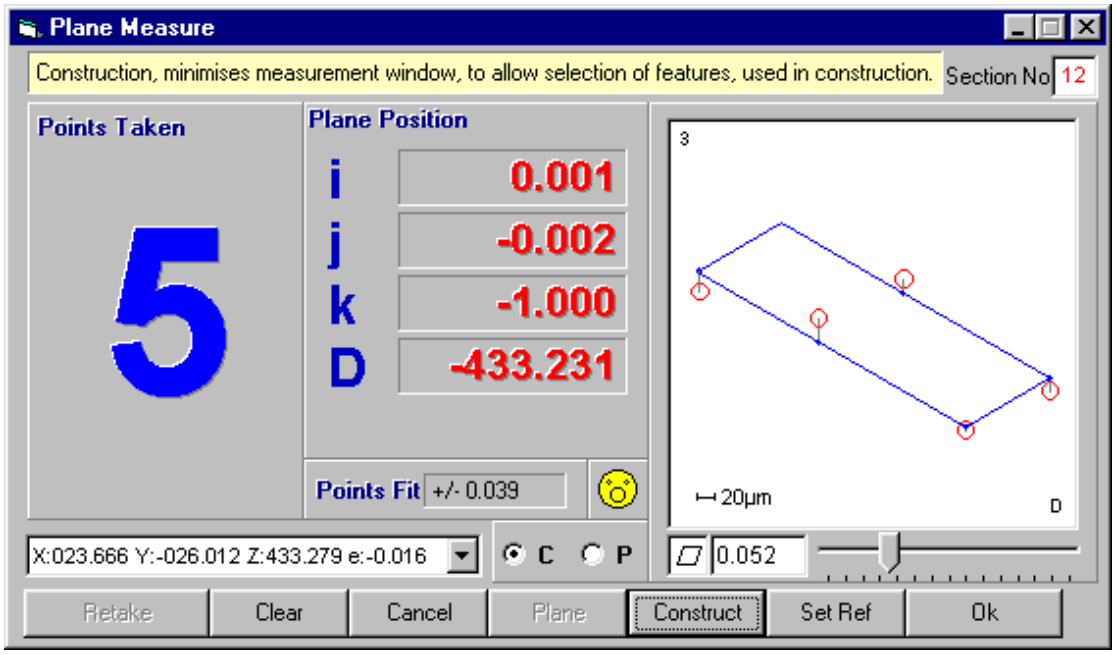
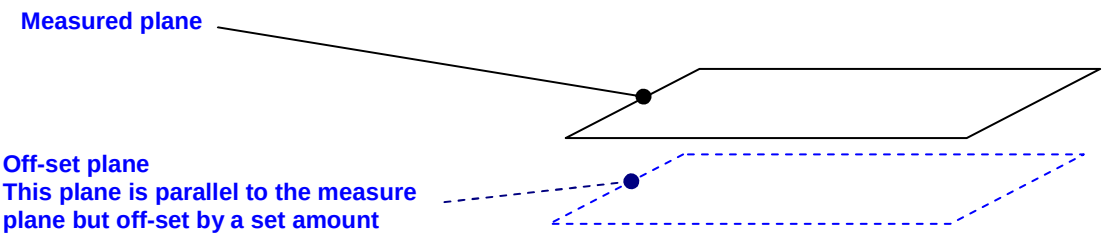


Fig. 8.5-3 A representation of a Plane which has been constructed

You may now click on 'OK' or 'Set Ref' as appropriate, as for any plane. The software will now return to the Main Screen.

www.EngineeringBooksPdf.com

8.6 Constructing a Plane offset from another Plane



A plane may be constructed which is offset from an existing plane by a fixed distance. To construct such a plane, click on the Plane Measure button, from the main screen. This will bring up the Plane Measure window. Now click on the 'Construct' box.

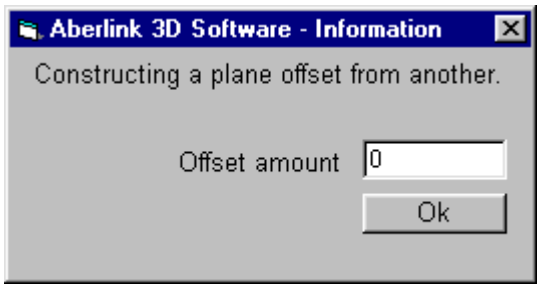


Fig. 8.6-1 An 'Offset Plane' Prompt

The Plane measure window will now shrink to a small box at the bottom of the screen. Now simply click on the plane from which you wish to construct an offset plane. A prompt will appear on the screen confirming what is happening, and inviting you to enter the offset distance.

Enter the desired offset, and then click 'OK'. This will bring back the Plane Measure window. Click 'OK' again. The screen will now return to the Main Screen, with the new plane shown offset from the original by the entered amount.

www.EngineeringBooksPdf.com

8.7 Constructing a mid plane between two measured planes

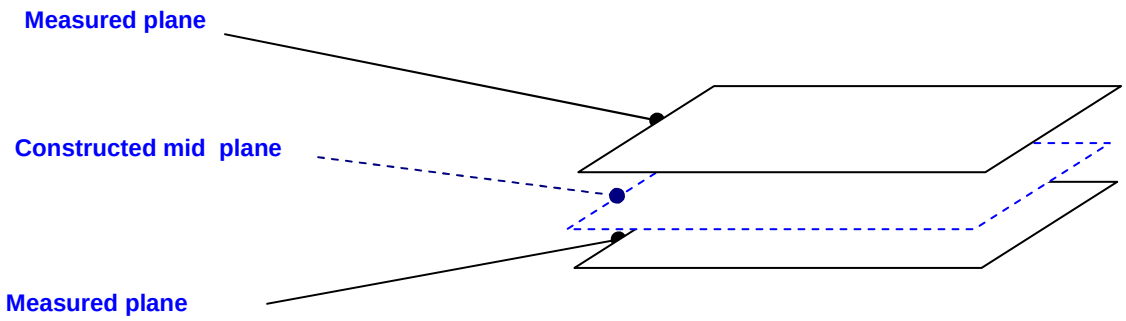


Fig. 8.7-1 constructed mid plane

It is possible to construct a plane midway between two other planes.
To do this you need to click on the 'Plane Measure button', from the main screen. This will bring up the Plane Measure window.
Now click on the 'Construct' button. The Plane measure window will now shrink to a small box at the bottom of the screen:

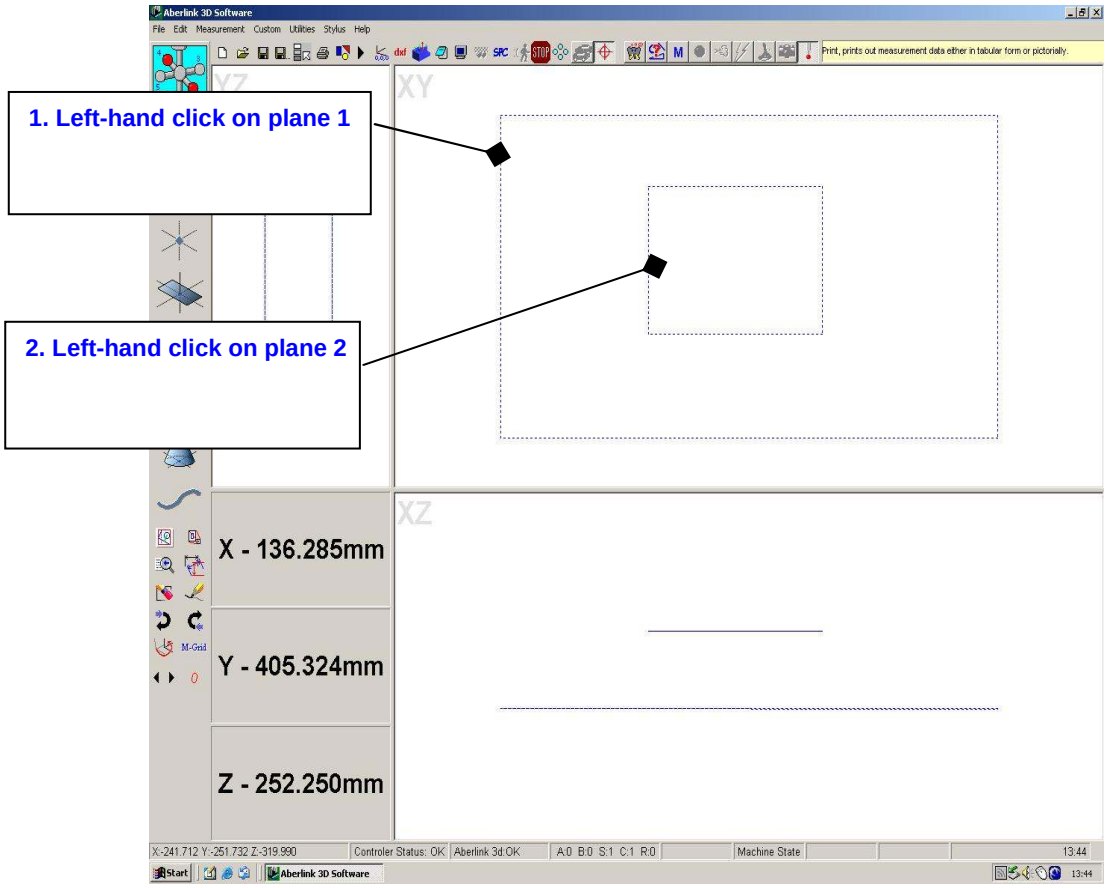


Fig. 8.7-2 selecting the planes

www.EngineeringBooksPdf.com

The next step is to left hand click on plane 1 then on plane 2 after you have done this a prompt will appear on the screen confirming you are constructing an mid plane.

Fig. 8.5-3 mid plane prompt window

When you see the above prompt click 'OK' and you now have a construct plane midway between the two selected planes.

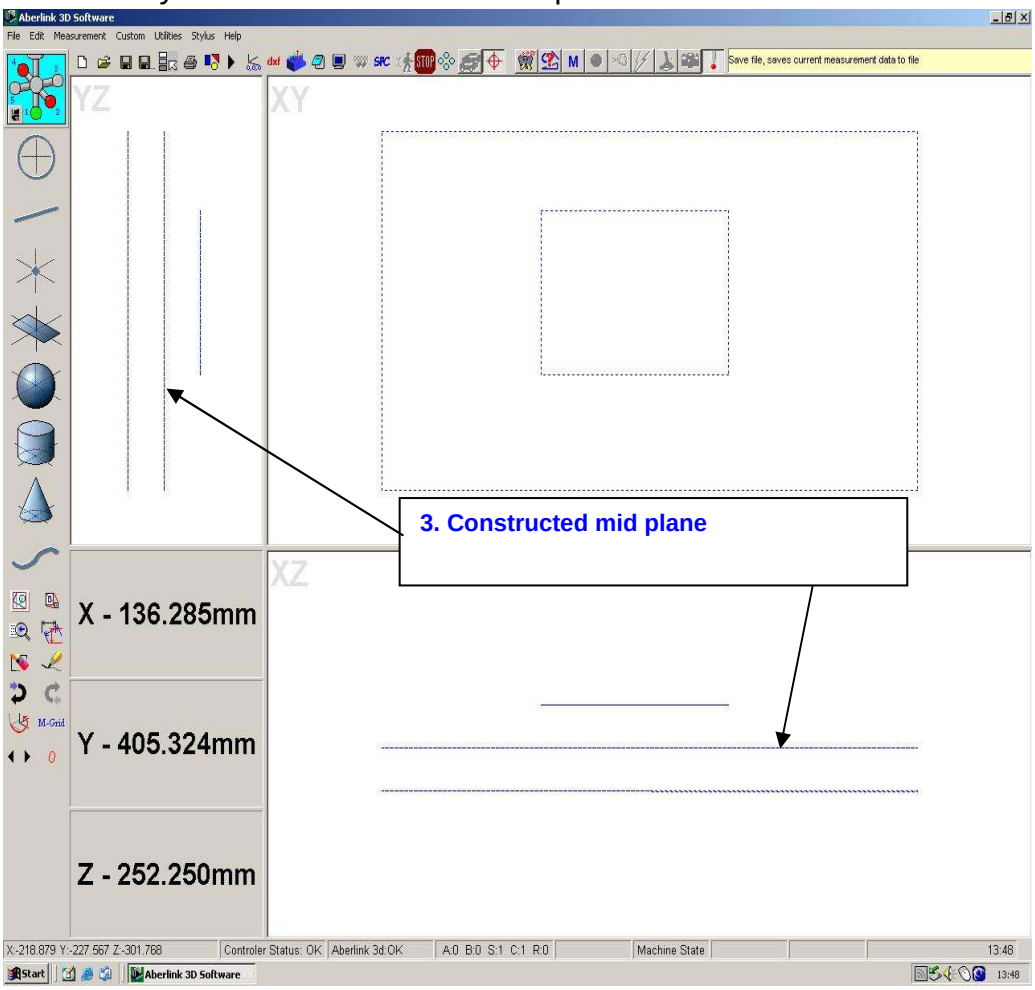


Fig. 8.7-4 constructed mid plane

www.EngineeringBooksPdf.com

9.0 THE LINE UNIT

The line unit is a very quick and accurate method of measuring flat faces, by taking 2 or more points along the length of the face, and projecting them into a defined plane to produce a 2 dimensional line.

- Lines may also be constructed through 2 or more previously measured points. In this case the line will have 3 dimensional properties.
- Lines can also be construct at the intersection of 2 non-parallel planes. Again this line will have 3 dimensional properties.
- You can also construct a tangent line between two circles or a point and a circle
- It is also possible to construct a perpendicular gauge line of a fixed length between too two other lines.
- Finally, It is also possible to construct a line that is the centre line between 2 other lines.

9.1 Measuring Lines Directly

Background
When measured directly, lines are 2 dimensional features that are used to represent flat faces, which by their nature are 3 dimensional features. This is achieved by projecting the measurement points taken (at whatever height) on to a defined plane to produce a straight line.

The software will decide whether the line best fits an XY, XZ or YZ plane, by looking at the direction of motion of the probe when the points are taken, If the plane of the line has previously been defined, then the position of the line can be calculated correctly, and its vector values will be displayed in red. If the plane has not been defined, then the machine XY, XZ or YZ plane will be used, which could lead to errors which may be significant. In this case the vector values of the line will be displayed in a light grey colour, by way of a warning.

Usually a plane is defined by setting it as the reference plane, but to fully define the orientation and position of a component see chapter 6.0 ALIGNING THE AXES OF A COMPONENT
When measuring a line the software will automatically look for meaningful intersections with other lines, in order to produce a complete outline of the component being inspected. The line will extend itself by a factor which may be set in the software set up – see28.0 SOFTWARE SET UP, to meet other co-planar lines. If a line has not been extended at both ends, its length will not be meaningful, and hence the L value will be shown in grey. If however, a line

www.EngineeringBooksPdf.com

has been successfully extended at both ends, its length is now relevant and the L value will be displayed in red.

Method

To measure a line, click on the Line Measure button, from the main screen:

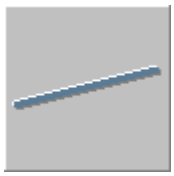


Fig. 9.1-1 The Line Measure button

This will bring up the Line Measure window:

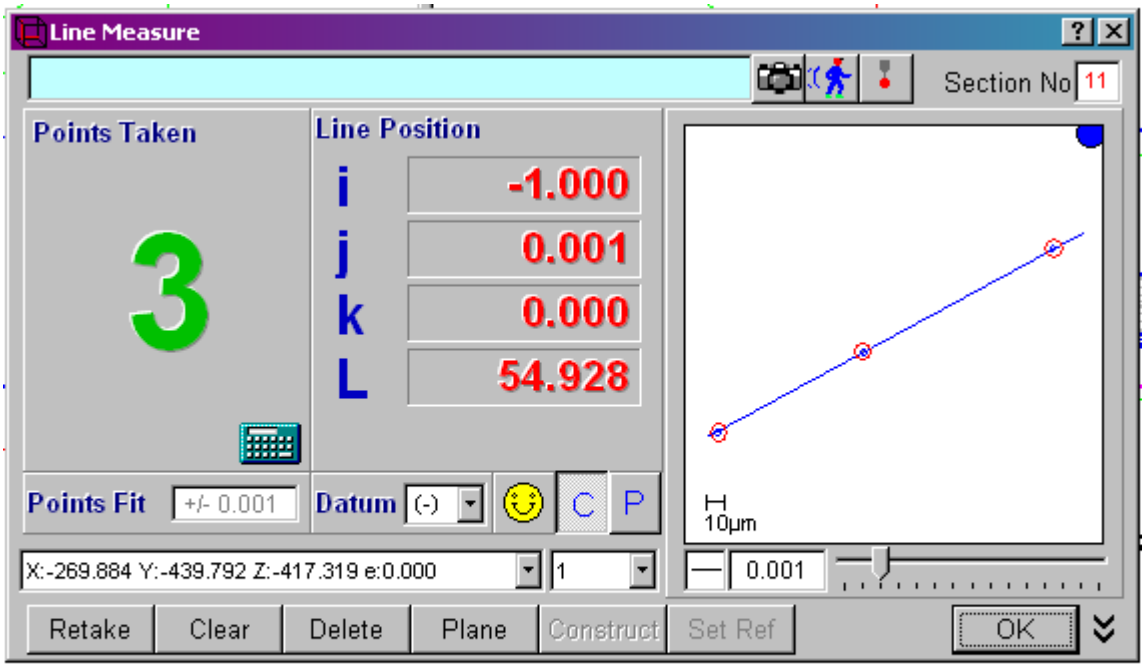


Fig. 9.1-2 The Line Measure window

Now take measurement points on the face that you wish to measure. After 2 points a pictorial representation of the line will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The I,J and K values displayed represent the direction vector for the line and the L value represents its length. The plane of the line is shown in the diagonal corners of the graphics window.

www.EngineeringBooksPdf.com

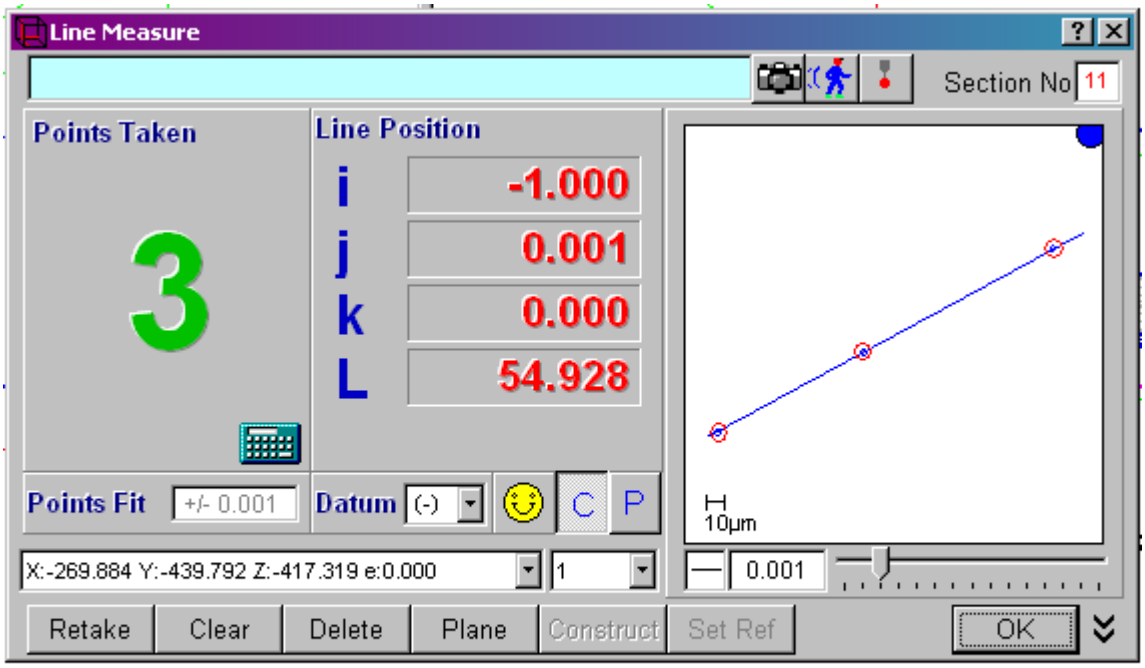


Fig. 9.1-3 The representation of a line through 3 measurement points

Note – That as long as a relevant plane has previously been defined, the I,J and K values will be shown in red. If these values are shown in grey, it is a warning that the software has had to use the machine's planes to calculate the position of the line, and if the component has not been aligned on the table could lead to errors.

Note also – That if a line is measured in isolation, the length given will be the distance between the extreme measurement points and will therefore be of little relevance. In this case the L value will be shown in grey. The software does, however, automatically extend lines to look for obvious intersections with other features. The extent to which a measured line can be extended can be set in the software set up. If a line has been successfully extended at both ends then the length value will be meaningful, and will be displayed in red.

A line can always be calculated to fit through two points, therefore the first 2 points will always produce single bleeps and the yellow face will remain smiling. Also the 'Points Fit' box next to the yellow face will remain blank and the straightness value for the line, shown under the graphical representation, will show zero.

It is therefore good practise to take at least one more point. On the third point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated 'best fit' line and a value for the straightness of this

www.EngineeringBooksPdf.com

line. Also there is now some meaning to the ‘bad point’ sound and face warnings. If the last point taken doesn’t fit the line very well, then the face will start to frown. If it doesn’t fit at all well, then the face will become sad, and the computer will make the ‘bad point’ sound. The threshold values for the smile and frown warnings can be user specified, see 28.1.5 Smile / Frown Threshold.

In general the more points taken on a line, the more information can be gleaned, and a minimum of 3 points is recommended. Similarly the points taken should cover as much of the face being measured as possible. The graphical representation of the line shows the points taken as a red circle with a blue line representing the ‘best fit’ line through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

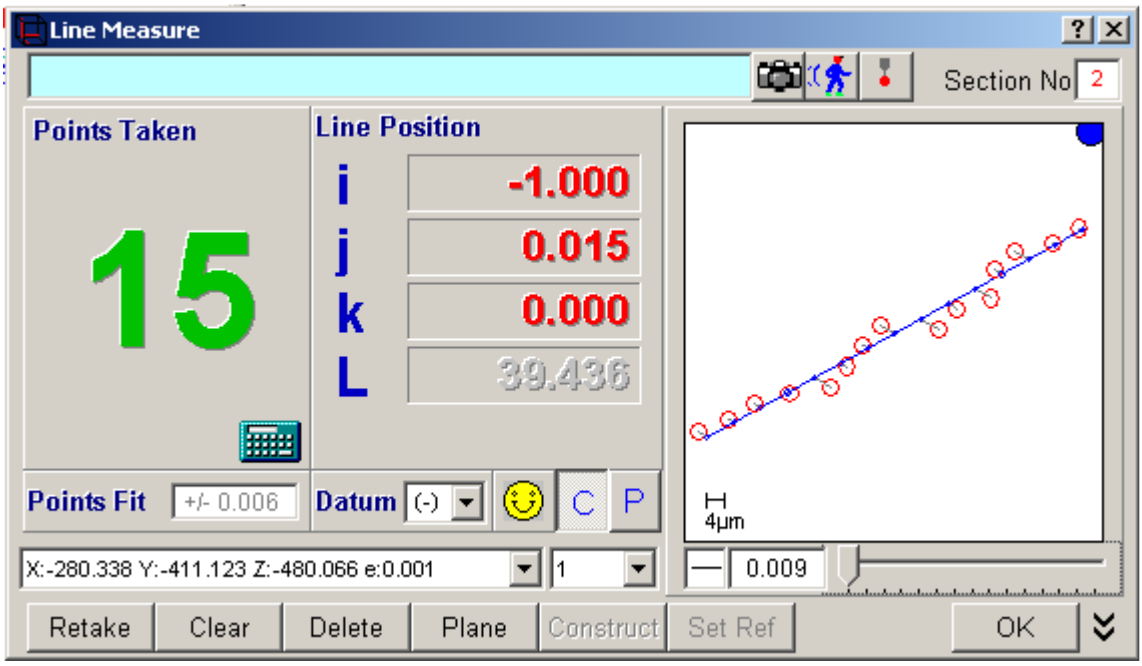


Fig. 9.1-4 A line with the error scale maximised

www.EngineeringBooksPdf.com

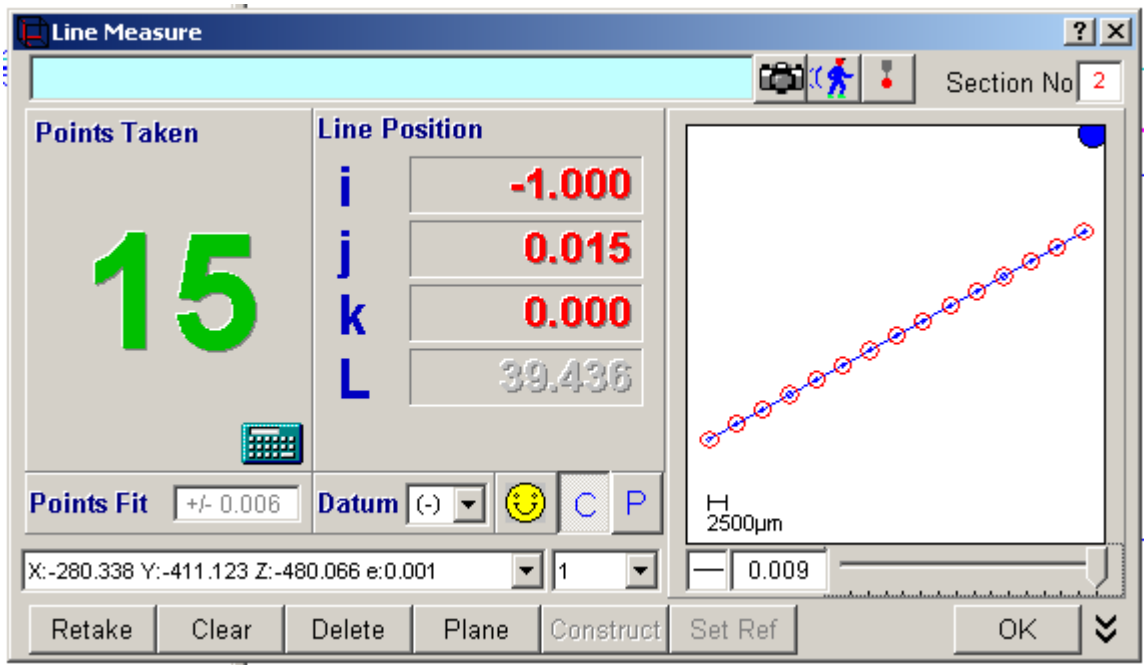


Fig. 9.1-5 line with the error scale minimised

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See chapter 13 - Erasing Data Points.

9.1.1 When to OK a line, and when to reference it.

Having measured a line, click on the '**OK**' button .

If you OK the line, the measurement window will close, and the program will return to the main screen.

If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position.

The graphical representation of the line should be shown as a solid blue line in all of the 3 views.

www.EngineeringBooksPdf.com

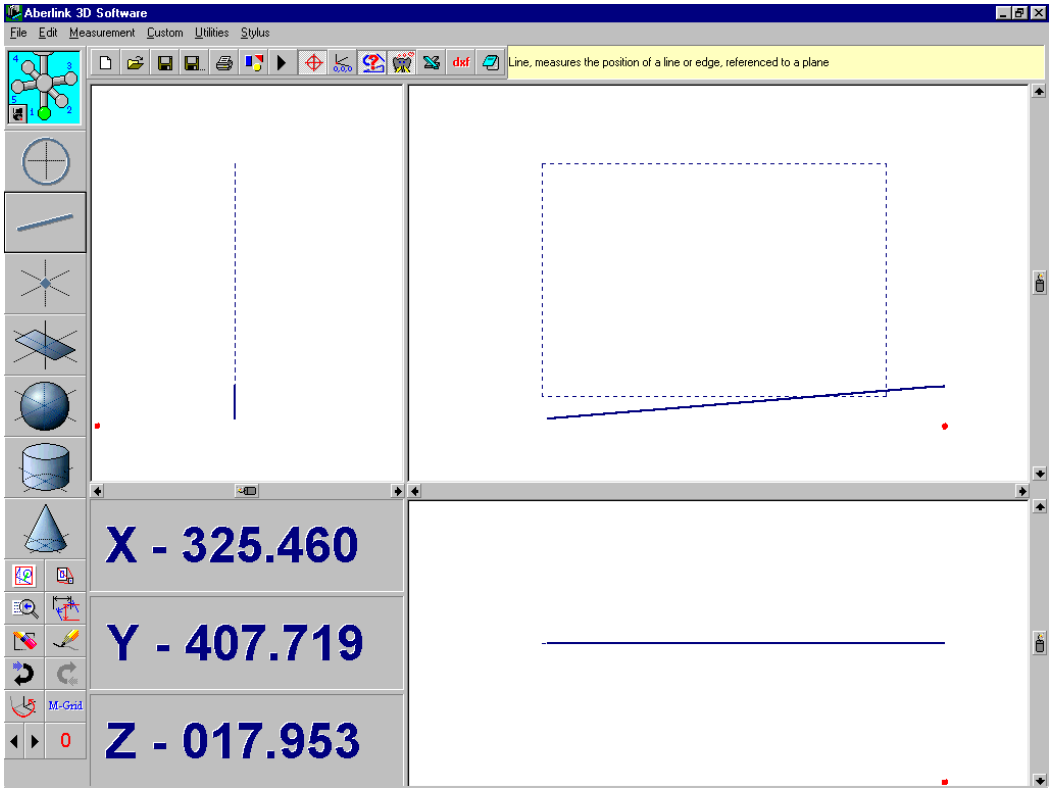


Fig. 9.1-6 The Main Screen showing a line that has been OK'd

If you set the line as a reference by clicking on '**Set Ref**' then the measurement window will say open until you click on the '**OK**' button the program will again return to the main screen.

The graphical representation of the line will once again be shown as a solid blue line within the reference plane in 2 of the 3 views, but this time the graphics picture in the third view will be aligned to the line itself. This line will now define the alignment, and position in one direction, of the reference plane. See 6.0 ALIGNING THE AXES OF A COMPONENT.

www.EngineeringBooksPdf.com

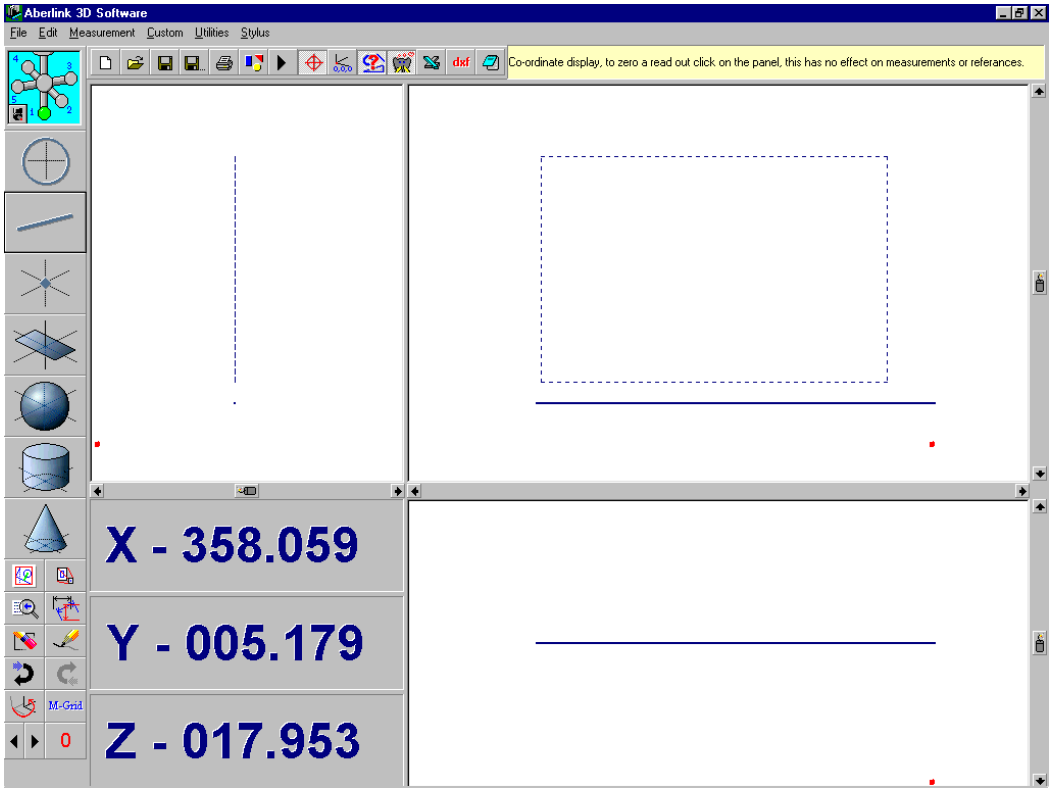


Fig. 9.1-6 The Main Screen showing a line that has been referenced

Note – Only one line can ever be set as a reference at any point in time. If a second line is set as a reference, this will override the initial reference line, and the graphical representation will now re-align itself to the second line. The first line that was referenced will now be treated as a reference point at its mid-point, and may be used to define the position of another plane.

9.1.2 Measuring a line in a user defined plane

If the software should have trouble in defining which plane a line is in, or if it is in a plane which cannot be defined by the orthogonal axes of the component, then it is still possible to measure the line, by the user defining the correct plane.

Having clicked on the Line Measure button to bring up the Line Measure window, click on the '**Plane**' button. This will bring up a window as follows:

www.EngineeringBooksPdf.com

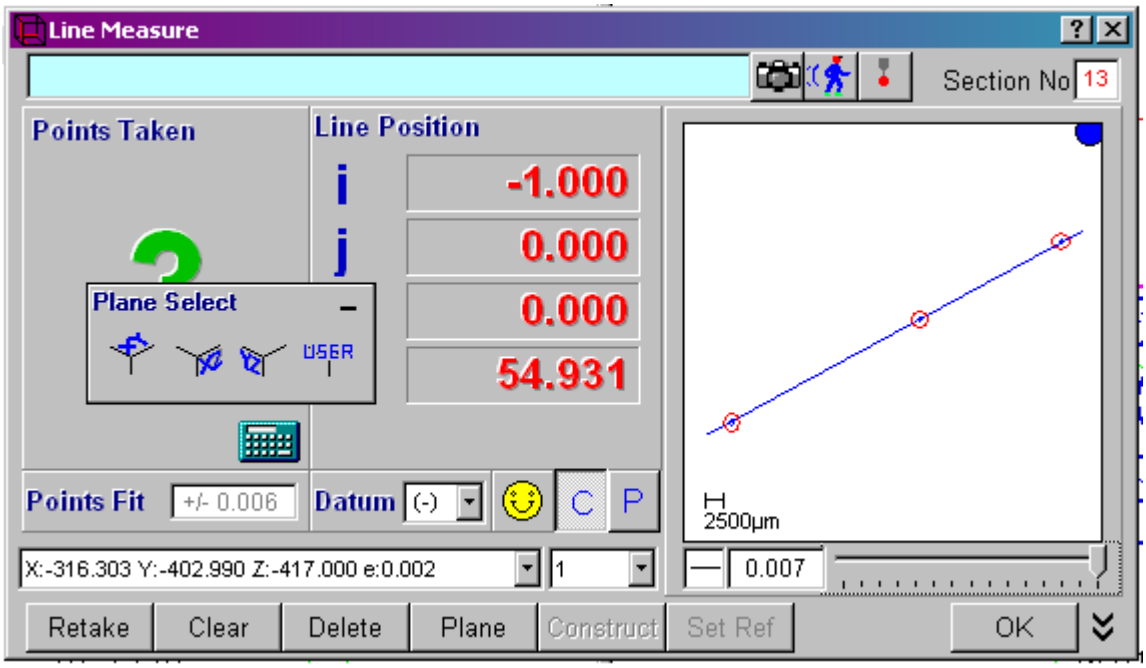


Fig. 9.1-8 The Line Measure window showing the Plane Select Feature

If you are simply defining the correct orthogonal plane, just click on the appropriate XY,XZ or YZ button that defines the line in the direction that you require, and then continue with the line measurement. If the required plane is not one of the orthogonal planes, you can still measure the line, as long as the relevant plane has previously been measured. Now click on the '**User**' button, and the following prompt will appear:

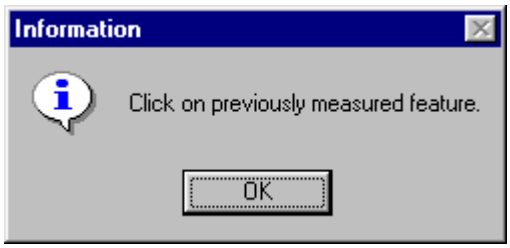


Fig. 9.1-9 The User Defined Plane prompt

Now simply click on '**OK**' and then click on the previously measured plane before continuing with the line measurement in the usual way.

www.EngineeringBooksPdf.com

9.2 Constructing Lines

9.2.1 Constructing Lines through 2 or more points

Background
Lines can be constructed through existing points in a very similar way to measuring then directly. The only difference is that instead of fitting a line through measurement points taken with the probe, you construct it through previously measured circles, points or spheres.

Method
To construct a line, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the '**Construct**' box. The Line Measure window will now shrink to a small box at the bottom of the screen.

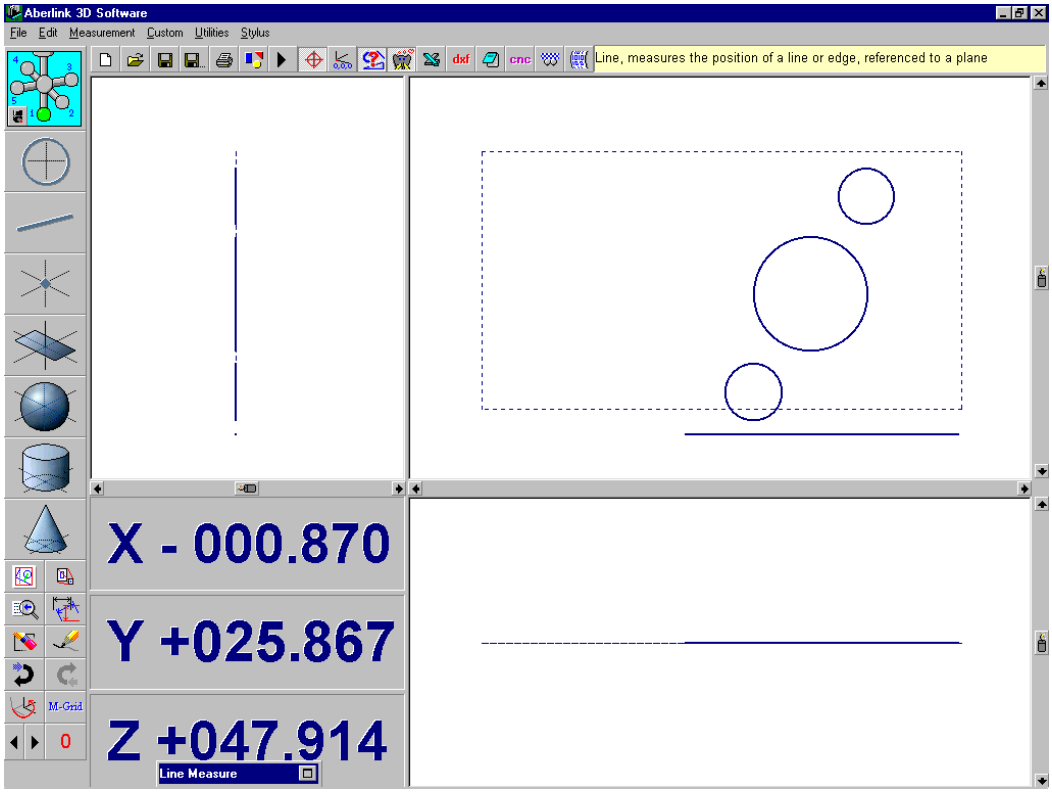


Fig. 9.2-1 The Main Screen showing a shrunken Line Measure window

Now simply select the features that you wish to use to construct the line by clicking on them. After you have selected the first feature a prompt will appear, to confirm what is happening:

www.EngineeringBooksPdf.com

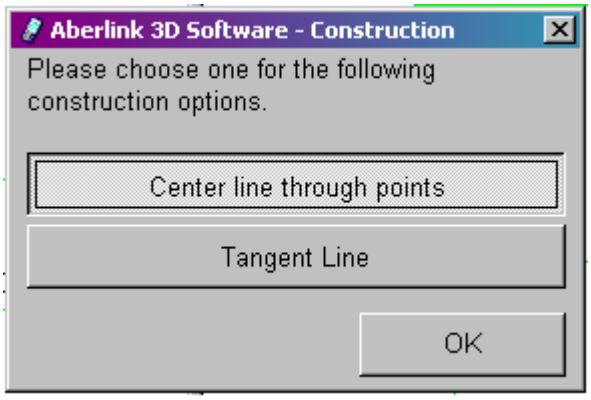


Fig. 9.2-2 The Constructing a Line prompt

Choose the **'Center line through points'** option then Click on **OK**, after this continue to click on the remaining features that make up the line. When you have selected all the desired features bring the Line Measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the line will appear with values for the fit and straightness given if more than 2 points were selected.

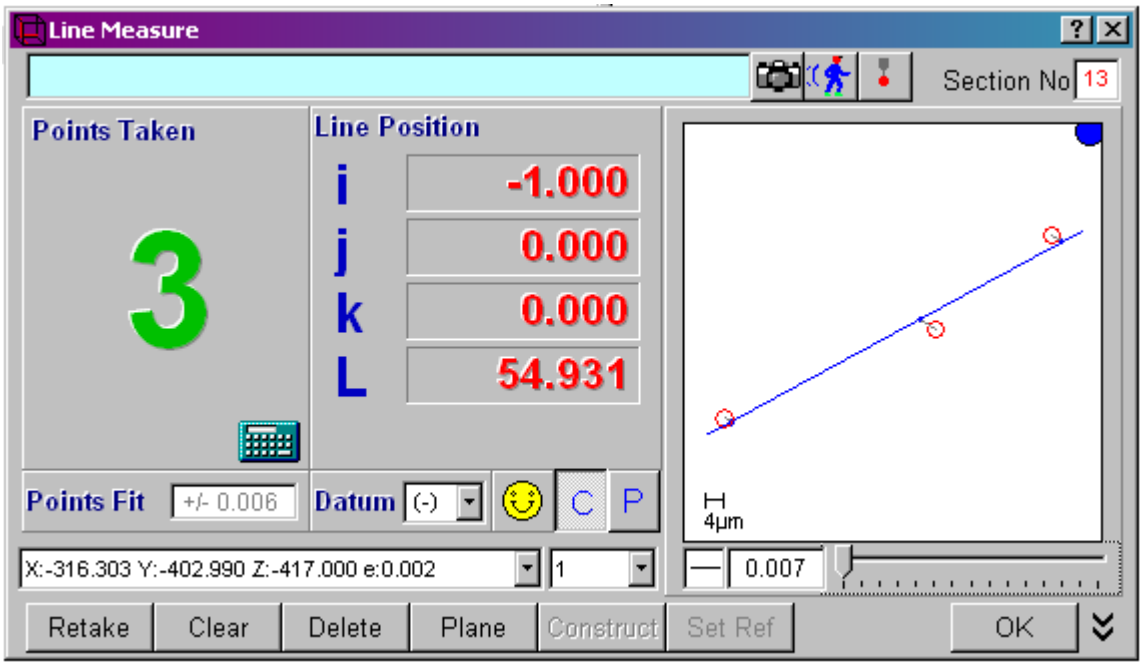


Fig. 9.2-3 Showing a line that has been constructed

You may now click on 'OK' or 'Set Ref' as appropriate, as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

9.2.2 Constructing a Line at the Intersection of 2 Planes

Background
In addition to constructing lines through existing points, lines can also be constructed at the theoretical intersection of two non-parallel planes.

www.EngineeringBooksPdf.com

Method

To construct a line at the intersection of two planes, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the 'Construct' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Now simply select the planes that you wish to use to construct the line by clicking on them. After you have selected the first plane a prompt will appear, to confirm what is happening:

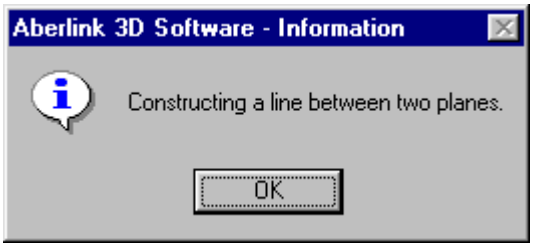


Fig. 9.2-4 The constructing the intersection between two planes prompt

Click on **OK**, and then click on the other plane that creates the desired line. The Line Measure window will reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:

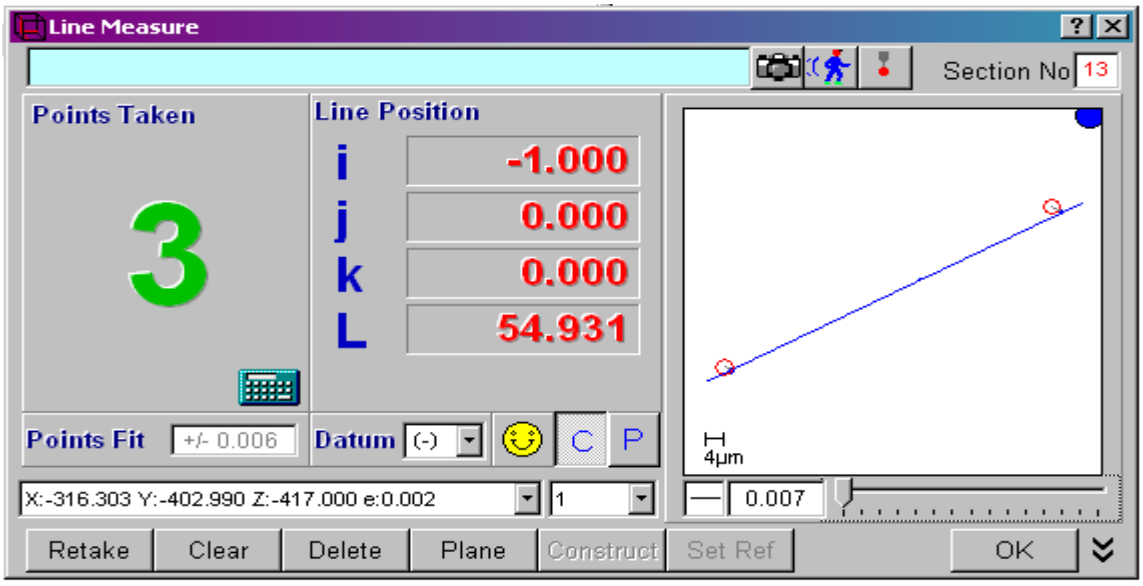


Fig. 9.2-5 line constructed between 2 planes

You may now click on '**OK**' as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

www.EngineeringBooksPdf.com

9.2.3 Constructing a Centre Line between 2 Other Lines

Background
One other line that can be constructed is a centre line between two other lines. If the 2 lines selected are not parallel, then the centre line constructed will bisect the angle between them.

Method
To construct a centre line between 2 other lines, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the 'Construct' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Now simply select the lines that you wish to use to construct the centre line by clicking on them. After you have selected the first line a prompt will appear, to confirm what is happening:

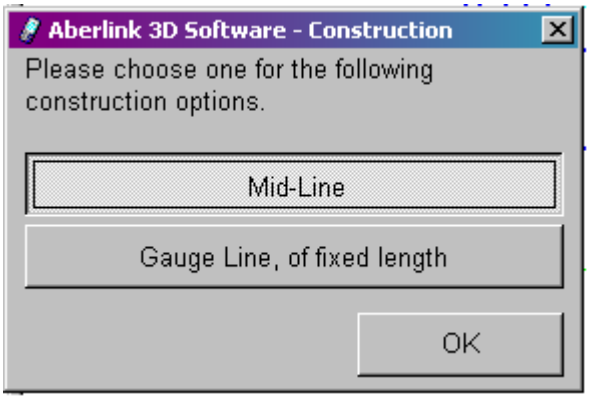


Fig. 9.2-6 The Constructing a centre line (Mid-Line) prompt

Choose the **'Mid-line'** option then Click on OK, and after this click on the other line that creates the desired centre line. The Line Measure window will reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:

www.EngineeringBooksPdf.com

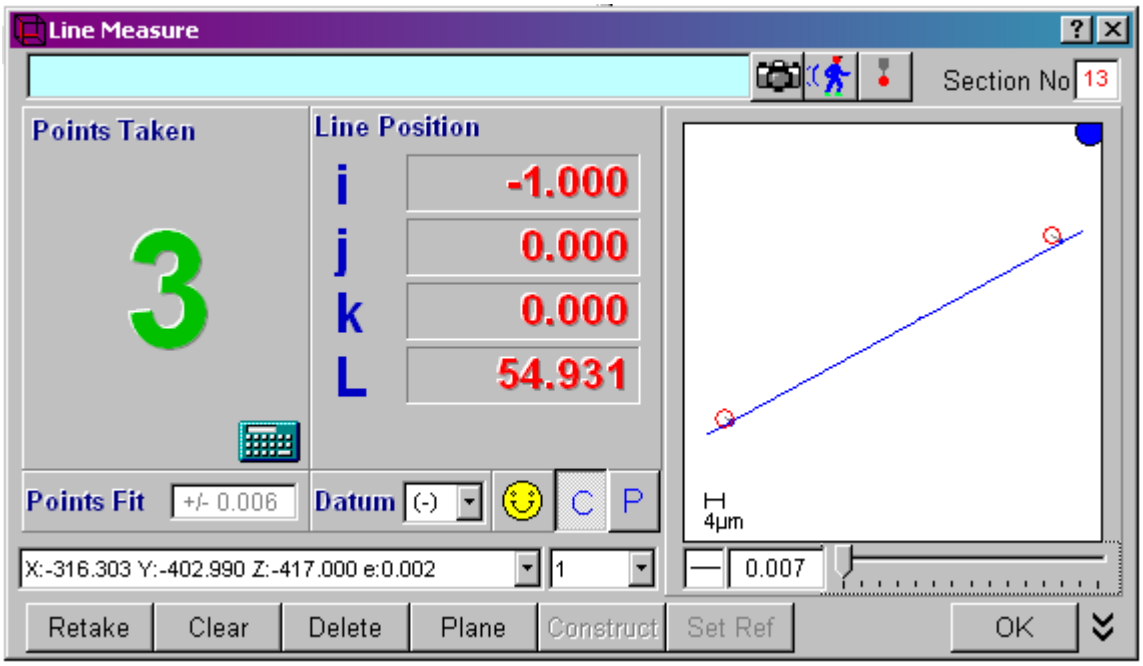


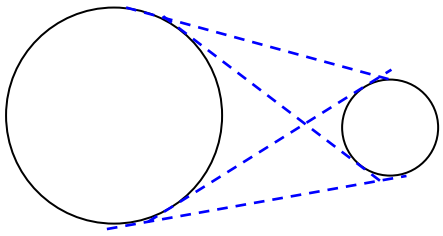
Fig. 9.2-7 A small line is show when constructing a centre line

You may now click on '**OK**' as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

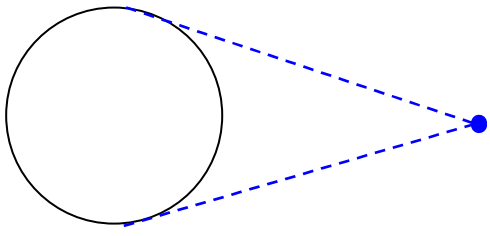
9.2.4 Constructing a Tangent Line between 2 circles or a point & circle

Background

With this function you can construct a tangential lines between two circles or between a point and a circle.



Tangent lines between two



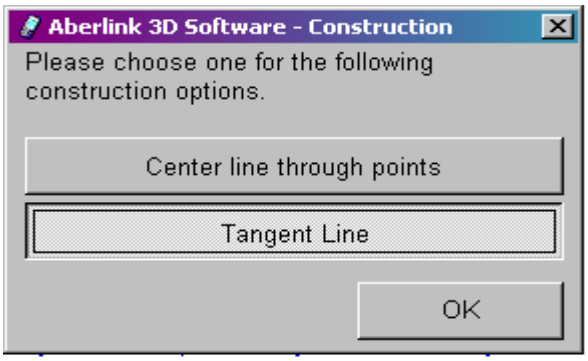
Tangent lines between a Point and a

Method

To construct a tangential line between two circles or between a point and a circle

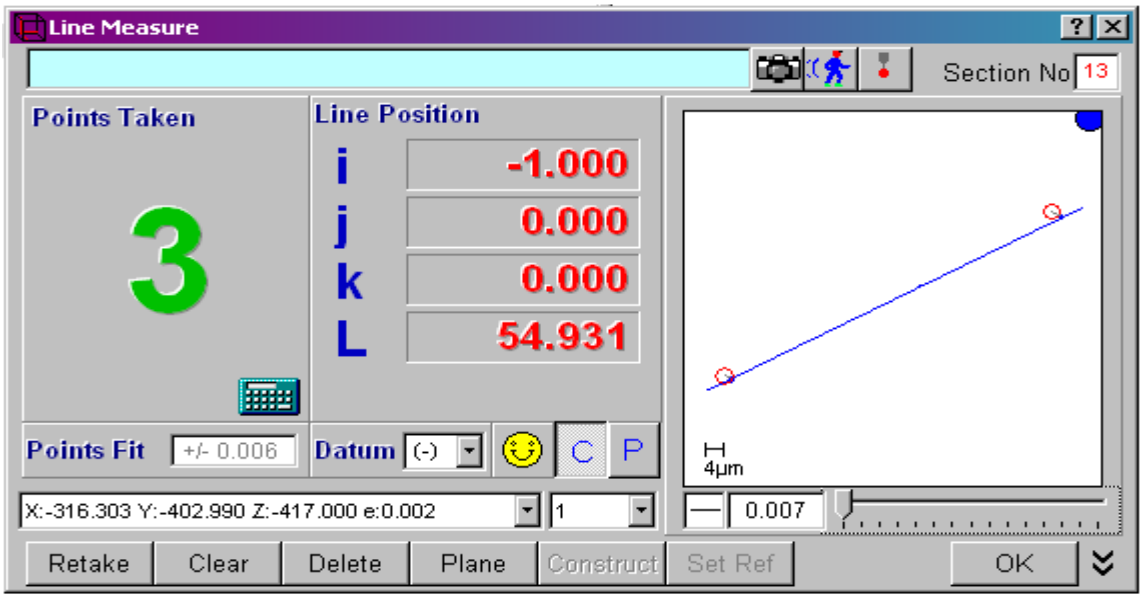
www.EngineeringBooksPdf.com

Click on the '**Line Measure button**', from the main screen. This will bring up the Line Measure window. Now click on the '**Construct**' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Note that between 2 circles there are actually 4 possible tangent lines that can be constructed, and 2 possible tangent lines between a point and a circle. Now simply select the first circle (or the point) followed by the second circle being careful to click on it at the position closest to where the desired tangent line will pass. The following prompt will then appear:



Choose the **'Tangent line'** option then Click on '**OK**'.

The Line Measure window will now reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:

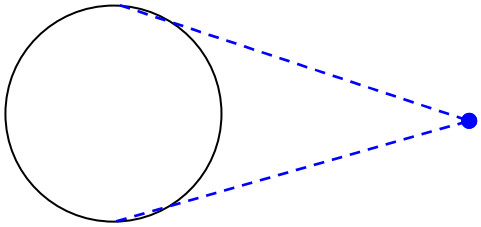


You may now click on '**OK**' as for any line. The screen will then return to the Main Screen, and the line will form a part of the graphical representation.

www.EngineeringBooksPdf.com

Example 1: ***constructing a tangent line between a point and a circle.***

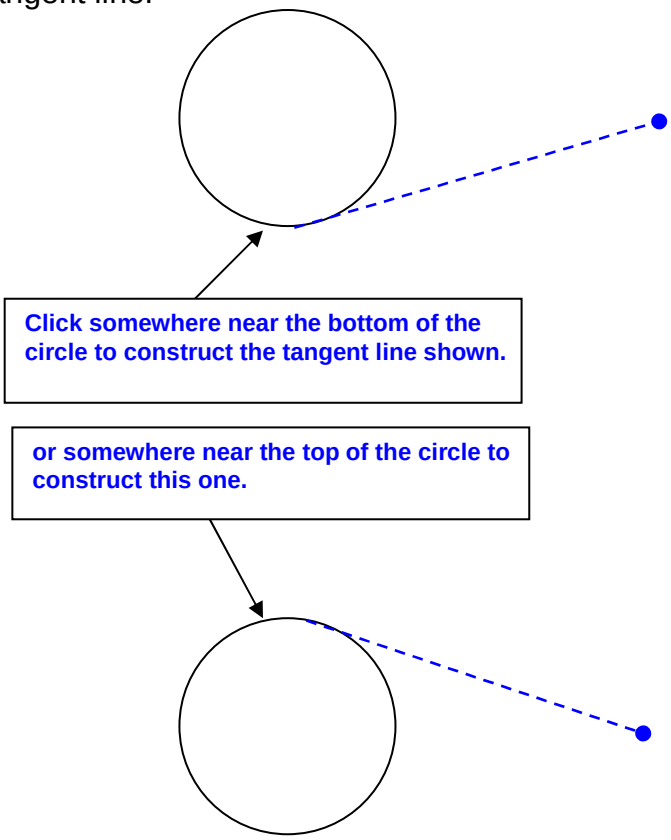
Note that there are 2 possible options, as follows:



1) Open a Line Measure Window and click on 'Construct' so that the window shrinks to a small box at the bottom of the screen.

2) Click on the Point to select it.

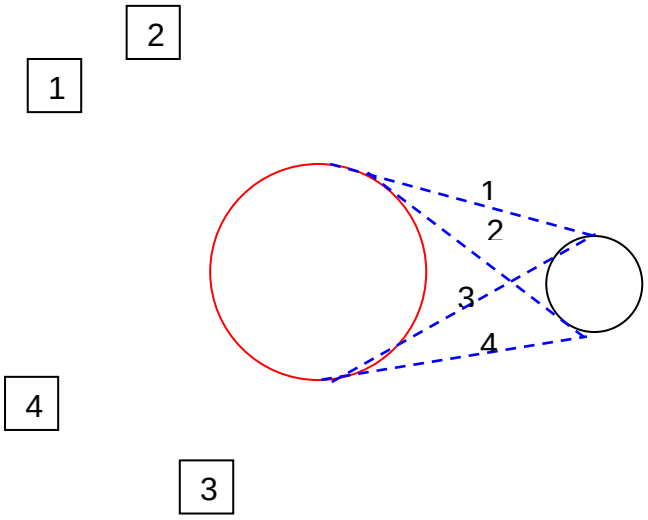
3) Now click on the circle at the appropriate position to choose the desired tangent line.



4) OK the Line Measure Window to return to the Main Screen.

www.EngineeringBooksPdf.com

Tip : Often the tangent lines between 2 circles can touch the circles at a very similar position making it difficult to choose the correct line required. In this case highlight the 2nd circle using the Feature Select buttons and click at a position that will clearly define the desired tangent line. For example, if you select the larger circle second, using the Feature Select function you can now click high up on the box labelled '2' in order to construct tangent line no2. Note that this position is through where tangent line no.2 will pass, and is clearly closer to no.2 than any of the others. Similarly the other boxes also show possible positions to click that will uniquely define the other tangent lines.



9.2.5 Constructing a Gauge Line of Fixed Length between 2 other Lines

Background

This function lets you construct a gauge perpendicular line of a fixed length between two other lines set at an angle to each other.

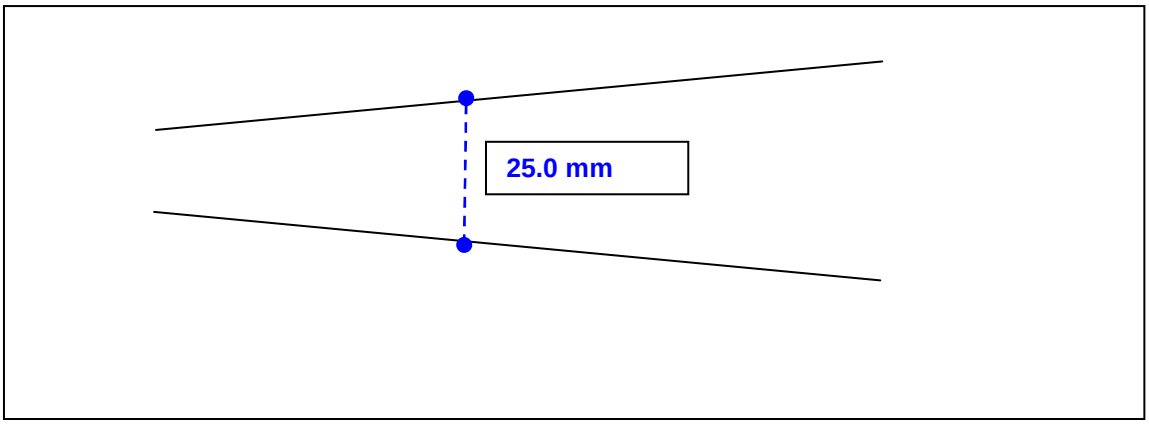


Fig. 7.2-11 Constructing a Gauge line of a fixed length

www.EngineeringBooksPdf.com

Method

To construct a perpendicular gauge line of a fixed length between 2 other lines, click on the Line Measure button, from the main screen. This will bring up the Line Measure window. Now click on the 'Construct' box. The Line Measure window will now shrink to a small box at the bottom of the screen. Now simply select the lines that you wish to use to construct the gauge to by clicking on them. After you have selected the first line a prompt will appear, to confirm what is happening:

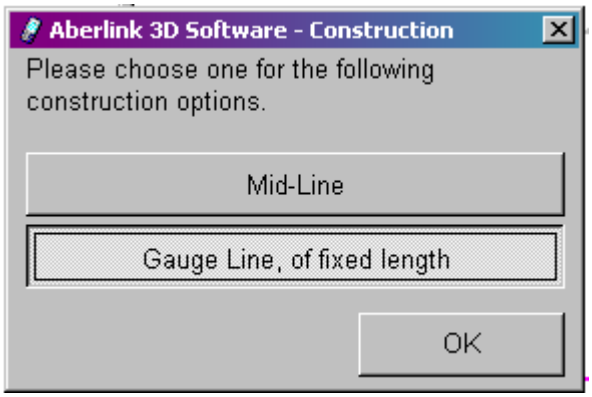


Fig. 9.2-12 The Constructing a gauge line prompt

Choose the **'Gauge line of fixed length'** option then Click on '**OK**', and after this click on the other line that you wish to connect the gauge line too. A prompt window will appear asking you for the length of the gauge line:

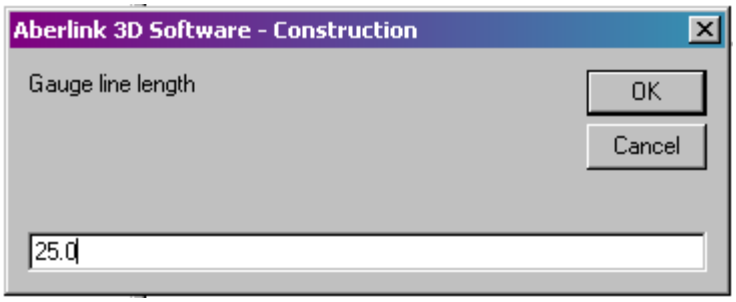


Fig. 9.2-13 The Constructing a gauge line size enter prompt

Enter the size of the gauge and click '**OK**' . The Line Measure window will now reappear. The number of points taken will be 2, and the graphics part of the window will show two points jointed with a line:

www.EngineeringBooksPdf.com

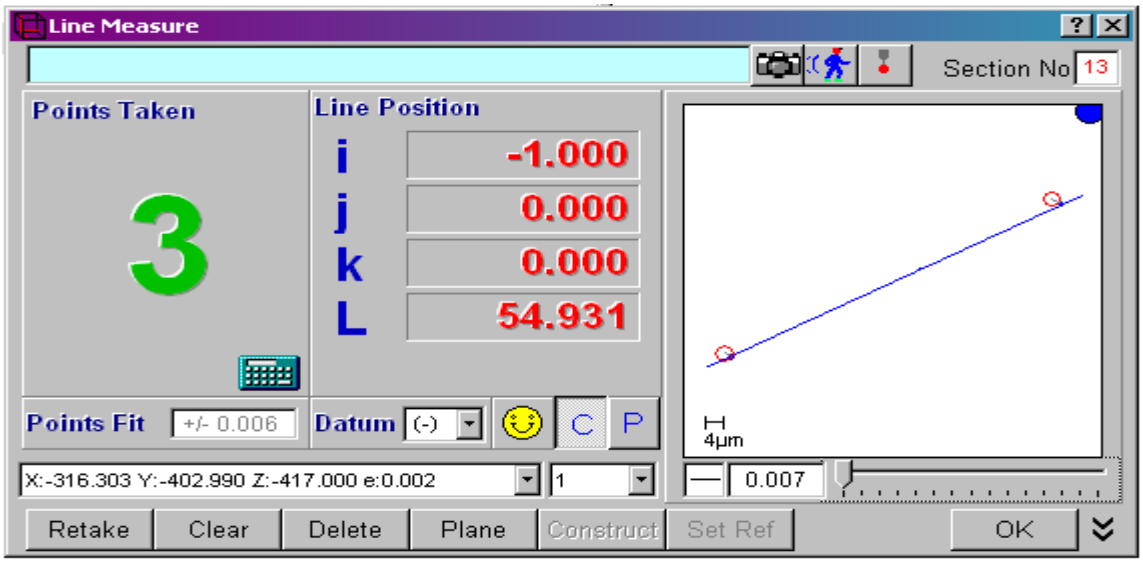
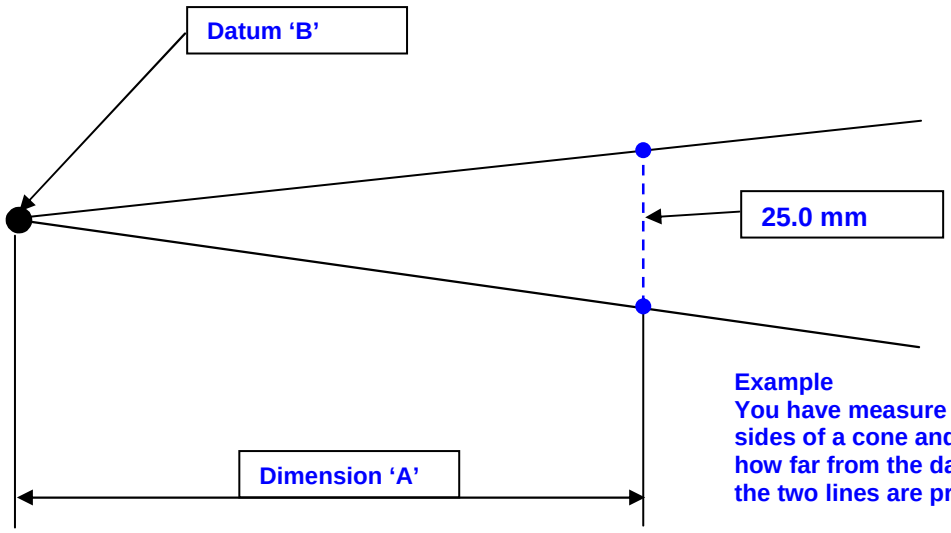


Fig. 9.2-12 A small line is show when constructing a gauge line

Now click on '**OK**' The screen will then return to the Main Screen, and the line will form a part of the graphical representation.



Example
You have measure two lines on opposite sides of a cone and you wish to know how far from the datum is the point were the two lines are precisely 25mm apart

www.EngineeringBooksPdf.com

10.0 THE SPHERE UNIT

Spheres can be measured directly by taking 4 or more measurement points on the surface of the sphere.

Background
The software will construct the 'best fit' sphere through the measurement points taken.

In order to determine whether it is an internal or external sphere, and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.

As a sphere is a 3 dimensional feature, no reference planes will need to be defined.

Method
To measure a sphere, click on the Sphere Measure button, from the main screen:

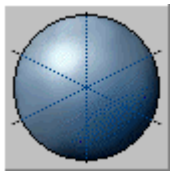


Fig. 10.0-1 The Sphere Measure button

This will bring up the Sphere Measure window:

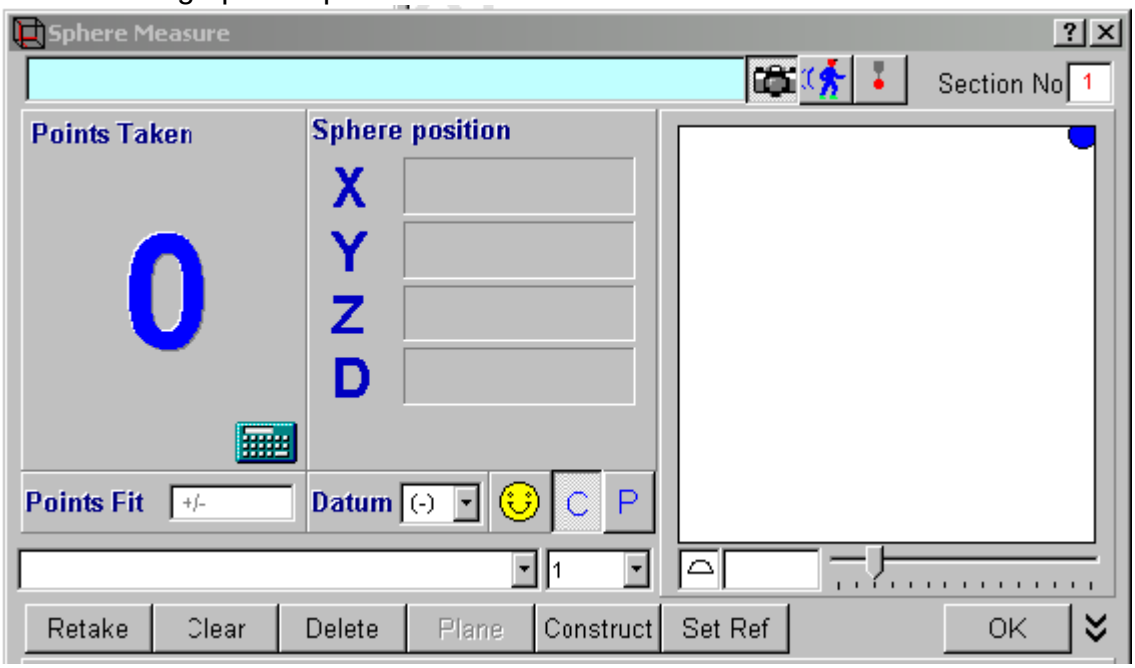


Fig. 10.0-2 The Sphere Measure window

Now take measurement points on the sphere that you wish to measure. After 4 points a pictorial representation of the sphere will appear in the graphics part of the window, which will be updated every point thereafter. The number

www.EngineeringBooksPdf.com

of points taken will be shown as the large character in the ‘Points Taken’ box. The X,Y and Z values displayed represent the centre of the sphere and the D value represents its diameter.

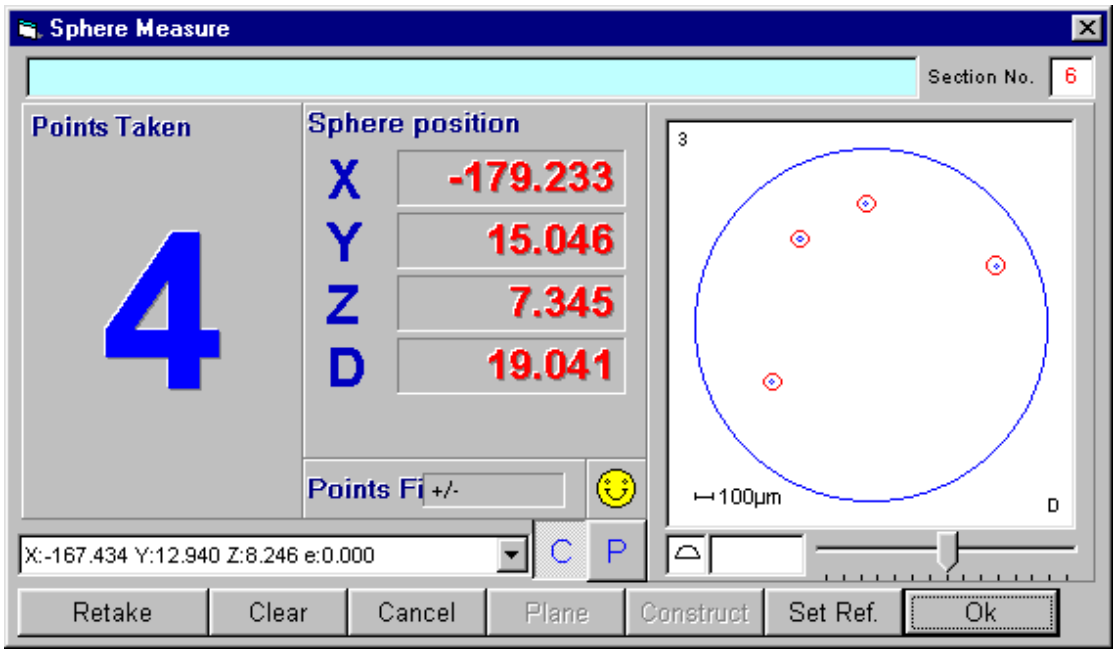


Fig.10.0-3 A graphical representation of the sphere will appear after 4 points

A sphere can always be calculated to fit through four points, therefore the first 4 points will always produce single bleeps and the yellow face will remain smiling. Also the ‘Points Fit’ box next to the yellow face will remain blank and the profile value for the sphere, shown under the graphical representation, will show zero.

It is therefore good practise to take at least one more point. On the fifth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated ‘best fit’ sphere and a value for the profile of this sphere. Also there is now some meaning to the ‘bad point’ sound and face warnings. If the last point taken doesn’t fit the sphere very well, then the face will start to frown. If it doesn’t fit at all well, then the face will become sad, and the computer will make the ‘bad point’ sound. The threshold values for the smile and frown warnings can be user specified, see 28.1.5 Smile / Frown Threshold

In general the more points taken on a sphere, the more information can be gleaned, and a minimum of 5 points is recommended. Similarly the points taken should cover as much of the surface of the sphere as possible.

The graphical representation of the sphere shows the points taken as a red circle with a blue line representing the best fit sphere through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

www.EngineeringBooksPdf.com

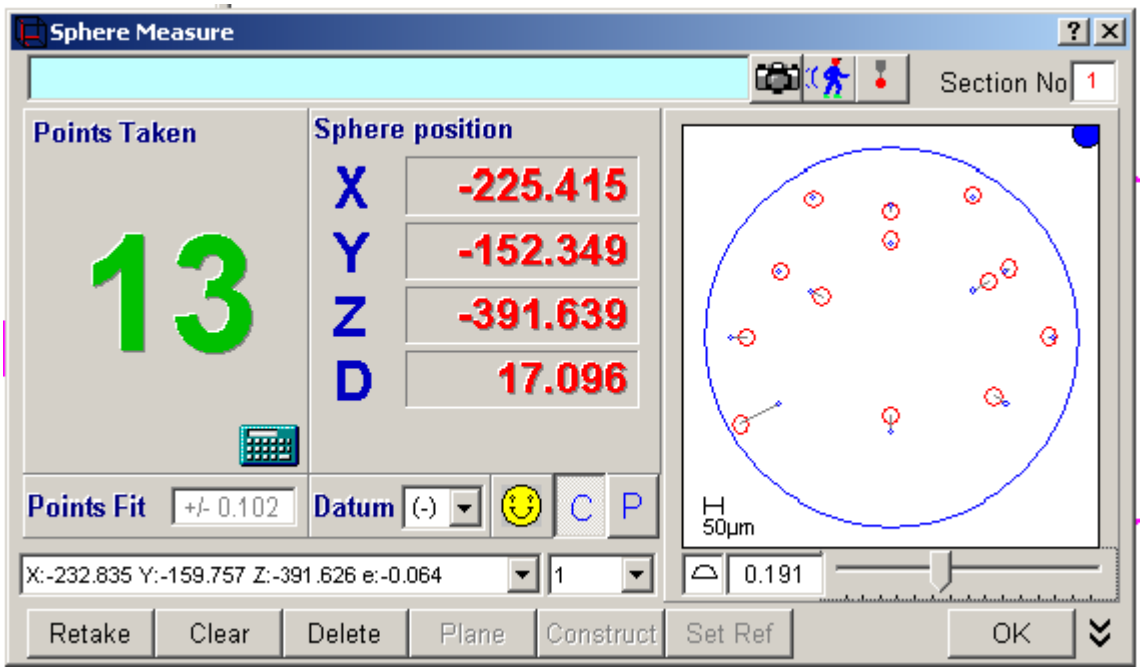


Fig. 10.0-4 A Sphere with the error scale maximised

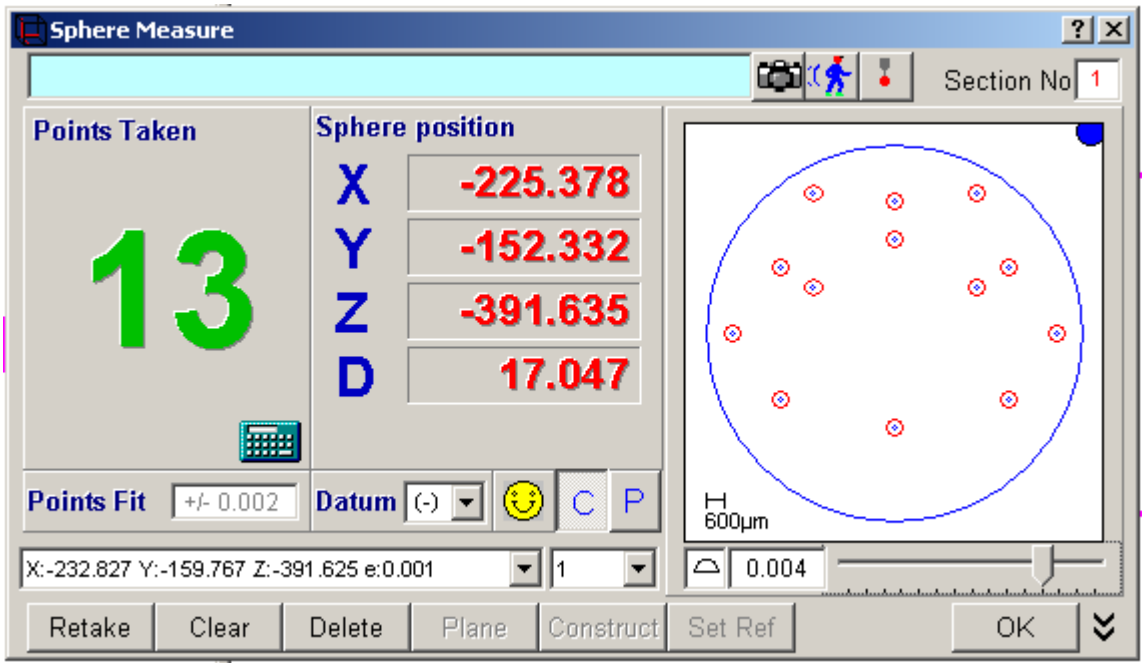


Fig. 10.0-5 A Sphere with the error scale minimised

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See chapter 13 - Erasing Data Points.

10.1 Displaying the centre position in polar co-ordinates

By default the centre co-ordinates of spheres are displayed as X,Y and Z values (carthesian co-ordinates). In order to display them in polar co-ordinate form, simply click on the ‘P’ button, situated below the ‘Point Fit’ box.

www.EngineeringBooksPdf.com

Now instead of displaying X,Y and Z values for the sphere, the screen will now display R, A and B values. This is now in polar co-ordinate form, with the R representing the distance from the X,Y origin, A being the angle in XY, and B being the elevation angle from the XY plane.

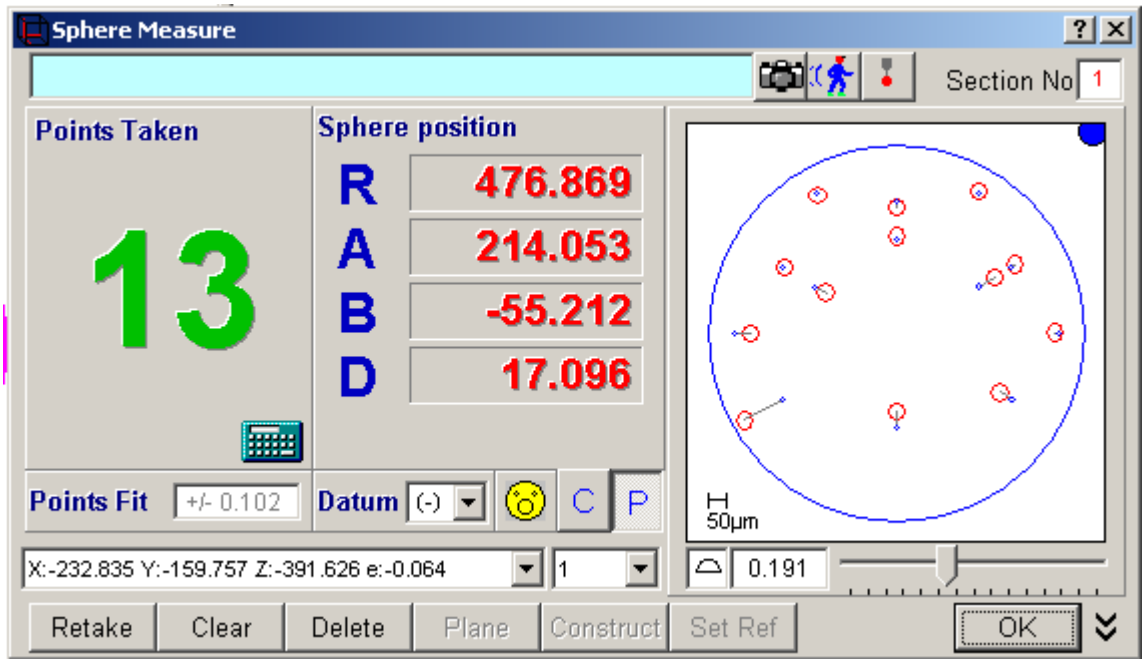


Fig. 10.1-1 A sphere described in polar co-ordinates

> ***Note*** *– that for these values to have any real meaning, a correct X,Y origin must have previously been set.*

To revert to carthesian co-ordinates simply click on the ‘C’ button.

10.2 When to OK a Sphere, and when to reference it.

Having measured a sphere, you can either click on the ‘OK’ button to close the sphere widow or click on the ‘Set Ref ‘ button to reference the cmm in the centre of the sphere .

The only effect that setting a sphere as a reference has, is to set it’s centre co-ordinates to X=0, Y=0, Z=0.

If you select ‘OK’ the centre co-ordinates of the sphere will be relative to any references previously set, or if none have been set then to the machine’s own reference point.

> ***Note*** *– Only one sphere (or point feature) can ever be set as a reference at any one time. If a second sphere (or another point) is set as a reference, this will override the initial reference point, and the centre co-ordinates of the second sphere will be set to 0,0,0.*

When you click on either ‘OK’ , the measurement window will close, and the program will return to the main screen. If any, or all of the graphics screens

www.EngineeringBooksPdf.com

are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position. See 17.0 MANIPULATING THE GRAPHICS.

www.EngineeringBooksPdf.com

11.0 THE CIRCLE UNIT

The circle unit is a very quick and accurate method of directly measuring holes, round bosses or arcs by taking 3 or more measurement points around the circumference.

Circles can also be constructed through 3 or more previously measured circles or points to create a pitch circle diameter.

In addition, circles can also be constructed at the intersection of various 3-dimensional features, such as the intersection of a plane with a perpendicular cylinder.

11.1 Measuring Circles Directly

Background
Circles are 2 dimensional features that are used to measure holes, bosses or arcs, which by their nature are 3 dimensional features. This is achieved by projecting the measurement points taken (at whatever height) on to a defined plane to produce a 2 dimensional circle.

The software will decide whether the circle best fits an XY, XZ or YZ plane, by looking at the direction of motion of the probe when the points are taken, (unless a user plane has been defined - see section 8.1.4). If the plane of the circle has previously been defined, then the position and size of the circle can be calculated correctly, and its centre co-ordinates and diameter will be displayed in red.
If the plane has not been defined, a warning window will be displayed telling you ‘The plane this feature is in can not be determined ‘ if you click ‘OK’ the machine XY, XZ or YZ plane will be used as the projected plane, which could lead to significant errors.

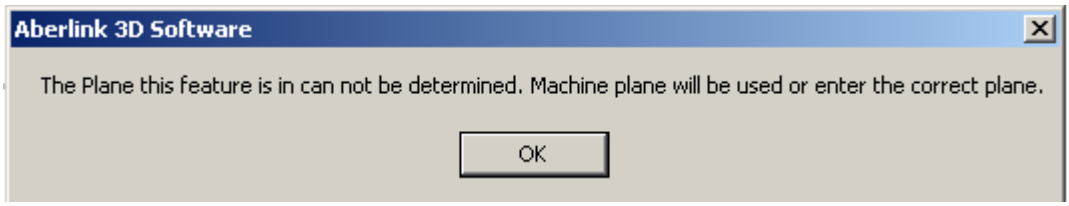


Fig. 11.1-1 The plane in which this feature be in can not be determined warning window

Usually a plane is defined by setting it as the reference plane, but to fully define the orientation and position of a component see chapter 6.0 ALIGNING THE AXES OF A COMPONENT

If the circular feature consists of less than 180 degrees, then the software will describe an arc between the extreme points. As soon as the measurement points span more than 180 degrees then a full circle will be described.

www.EngineeringBooksPdf.com

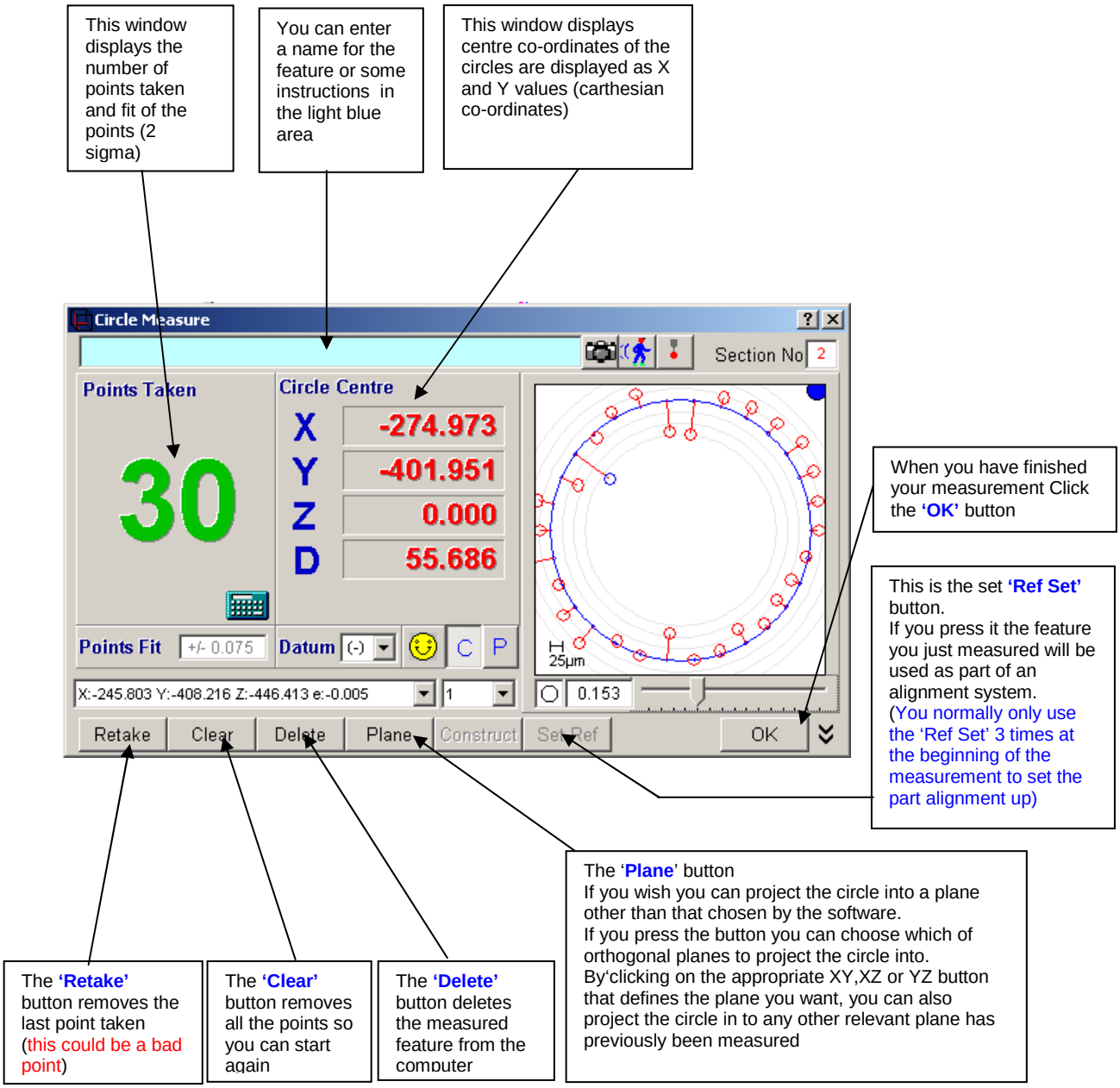


Fig. 11.1-2 The circle Window explained

Method

To measure a circle, click on the Circle Measure button, from the main screen:

www.EngineeringBooksPdf.com

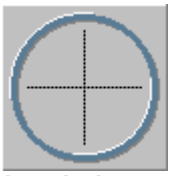


Fig. 11.1-3 The Circle Measure button

This will bring up the Circle Measure window:

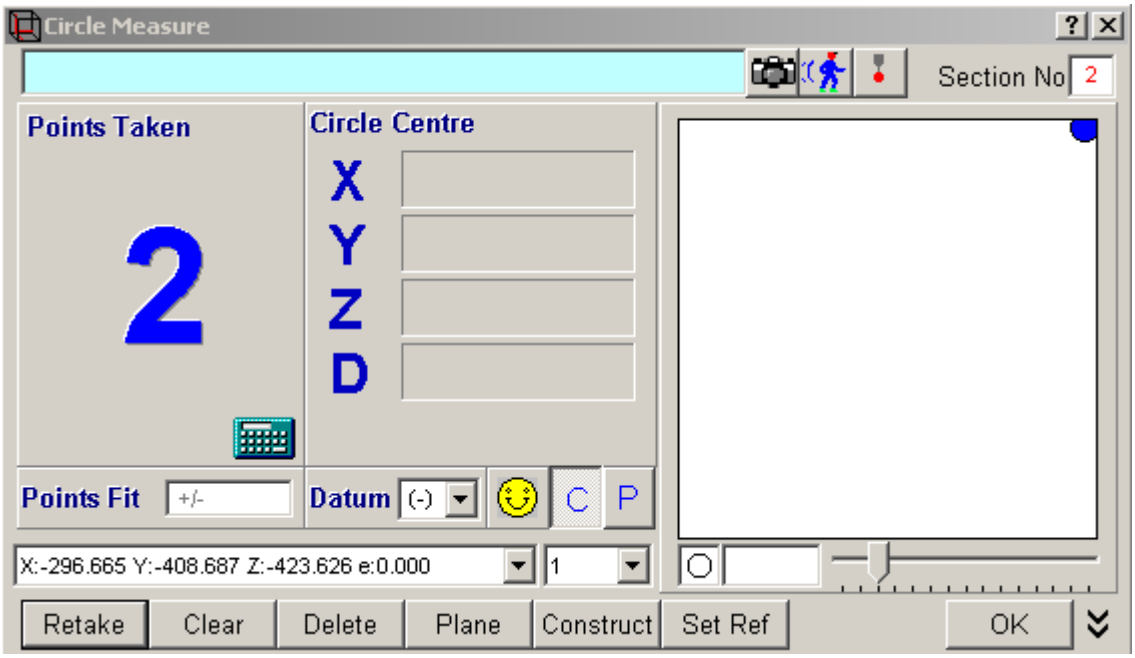


Fig. 11.1-4 The Circle Measure window

Now take measurement points on the hole or boss that you wish to measure. After 3 points a pictorial representation of the circle will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The X,Y and Z values displayed represent the centre of the circle and the D value represents its diameter.

www.EngineeringBooksPdf.com

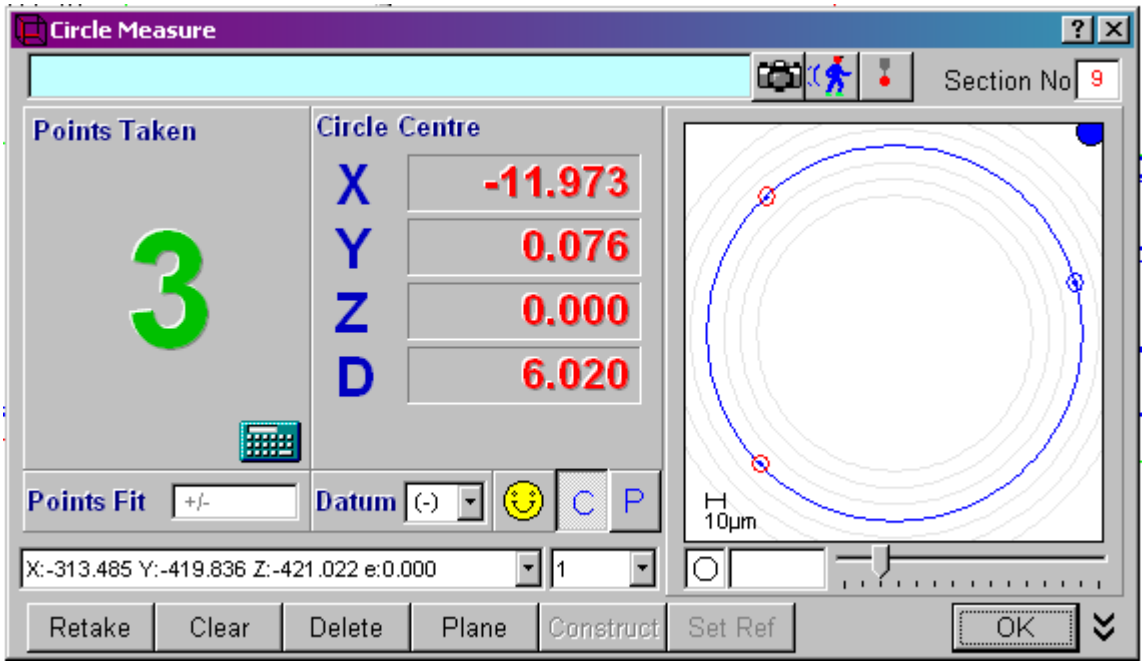


Fig. 11.1-5 After 3 measurement points a graphical representation will start to appear

> ***Note*** *– As long as a relevant plane has previously been set as a reference, the X,Y,Z and D values will be shown in red. If the warning window appears, it is telling you that either no reference plane has been set, or alternatively that the orientation or position of the circle has not fully been defined.*

A circle can always be calculated to fit through three points, therefore the first 3 points will always produce single bleeps and the yellow face will remain smiling. Also the 'Points Fit' box next to the yellow face will remain blank and the roundness value for the circle, shown under the graphical representation, will show zero.

It is therefore good practise to take at least one more point. On the fourth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated best fit circle and a value for the roundness of this circle (Rmax-Rmin. Also there is now some meaning to the 'bad point' sound and face warnings. If the last point taken doesn't fit the circle very well, then the face will start to frown. If it doesn't fit at all well, then the face will become sad, and the computer will make the 'bad point' sound. The threshold values for the smile and frown warnings can be user specified, see 28.1.5 Smile / Frown Threshold
In general the more points taken on a circle, the more information can be gleaned, and a minimum of 4 points is recommended. Similarly the points taken should cover as much of the circumference of the circle as possible.

The graphical representation of the circle shows the points taken as a red circle with a blue line representing the best fit circle through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

www.EngineeringBooksPdf.com

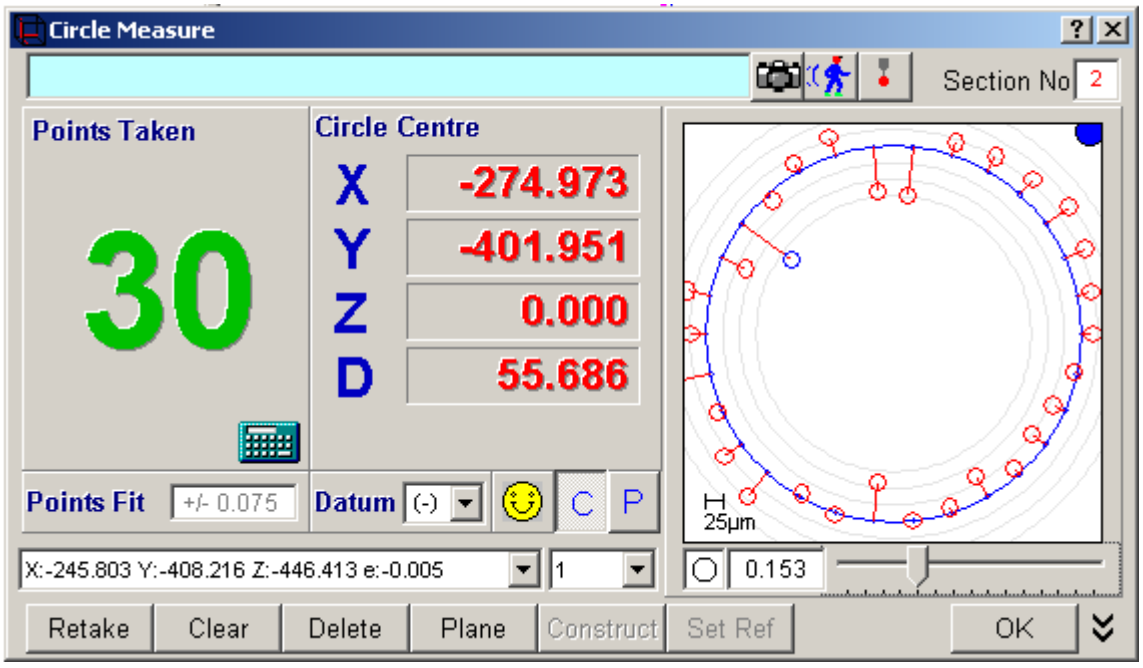


Fig. 11.1-6 A circle with the error scale maximised

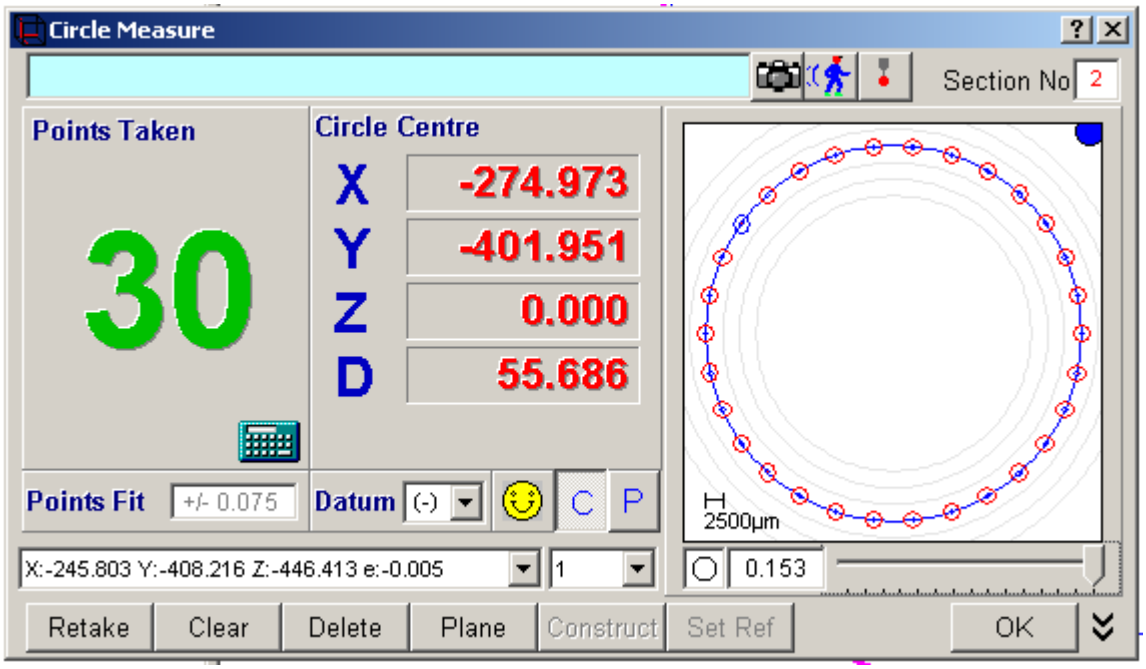


Fig. 11.1-7 A circle with the error scale minimised

If you have accidentally probed on the wrong place on the work piece, or are unhappy with a point, you can always erase either an individual point or all the stored points. See 16.0 ERASING DATA POINT.

www.EngineeringBooksPdf.com

11.1.1 Displaying the centre position in polar co-ordinates

By default the centre co-ordinates of the circles are displayed as X and Y values (carthesian co-ordinates). In order to display them in polar co-ordinate form, simply click on the ‘P’ button, situated below the ‘Point Fit’ box.

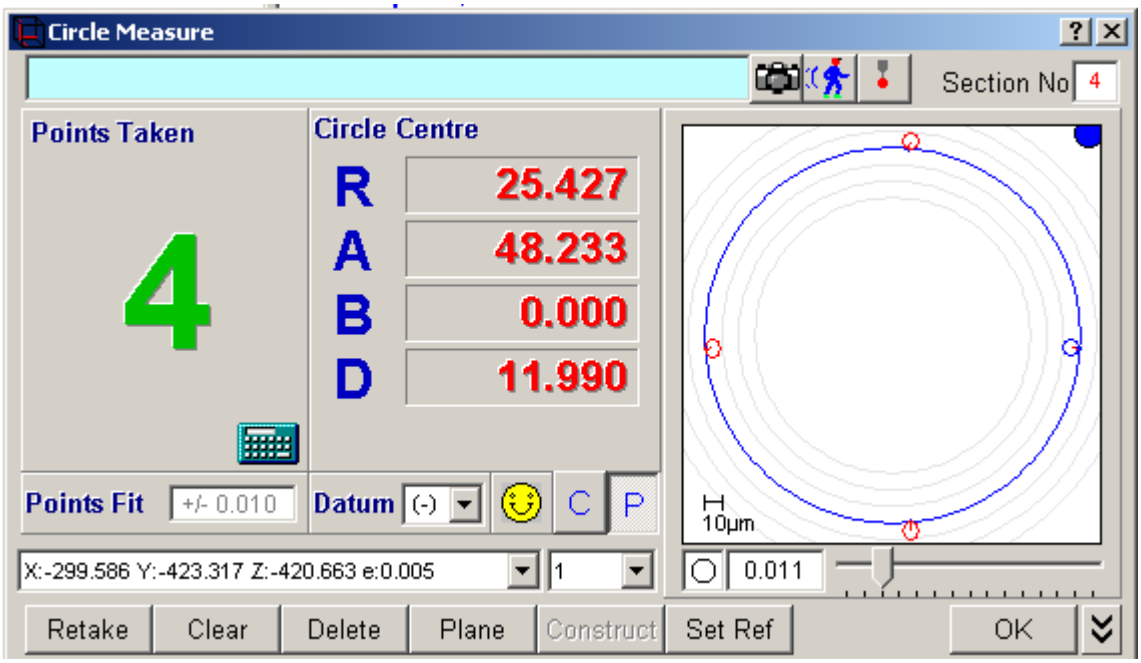


Fig. 11.1-8 A circle shown in polar co-ordinates

Now instead of displaying X,Y and Z values for the centre of the circle, the screen will now display R, A, and B values. This is now in polar co-ordinate form, with the R representing the distance from the X,Y origin, A being the angle in the XY plane, and B being the angle of elevation from the XY plane.

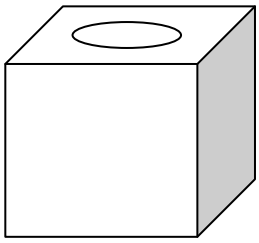
> *Note: – that for these values to have any real meaning, a correct X,Y origin must have previously been set.*

To revert to carthesian co-ordinates simply click on the ‘C’ button.

11.1.2 Off-line Programming.

Instead of directly measuring a circle using the probe it is possible to enter the co-ordinates for the centre of the circle and let the software calculate the measurement points to be taken around the circle. In this situation note that the points around the circle will be equally spaced, even if there are other features or obstacles in the way:

For instance we want to produce an off-line program for a circle at the centre



www.EngineeringBooksPdf.com

of the top face of a 40mm cube. Assume that the hole diameter in the centre of the top face is 25mm and that the top, front, left hand corner of the cube has been set as a reference, ie. is at X=0, Y=0, Z=0.

Click on the Circle Measure button, from the main screen to bring up the Circle Measure window. Now instead of taking points with the probe we are going to type in the template details directly:

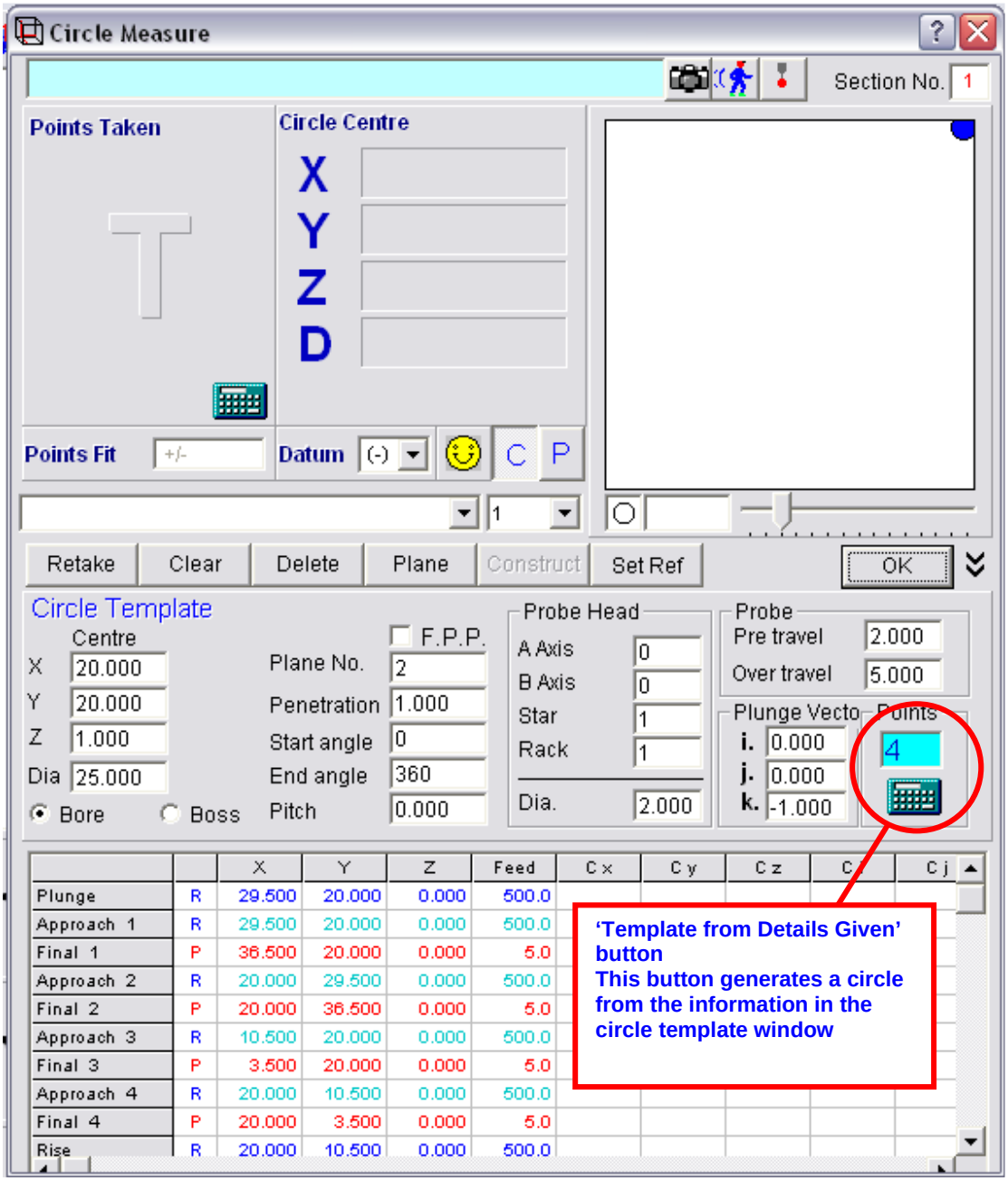


Fig. 11.1.2-1 'Template from Details button'

A circle is defined by entering the X, Y, Z position of the centre of the circle and its diameter. In this case the centre of the circle is at X=20, Y=20, Z=0 and the diameter is 25.

www.EngineeringBooksPdf.com

Where it says ‘CAL’ in the Plane No. box, it means that the software will automatically calculate the most appropriate plane for the circle.

Note also that although the circle is at Z=0 we have entered a penetration of 2mm, which means that the points will be taken with the centre of the ruby stylus at 2mm below the surface or Z=-2. This value defaults to the radius of the stylus being used.

The pre-travel and the over-travel will be set at their default values, but can be modified as required. Similarly you can change the probe head position and plunge vector (direction from which the probe will plunge prior to approaching the circle) if required. The default number of points on the circle has been set to 4.

Note that the ‘Bore’ or ‘Boss’ buttons will denote whether the circle is internal or external. In this case we want to measure a hole so the default ‘Bore’ is correct.

Now click on the ‘Template from Details Given’ button. The probe moves grid will now be automatically filled in, and the 4 points will be positioned equally spaced around the circle defined:

www.EngineeringBooksPdf.com

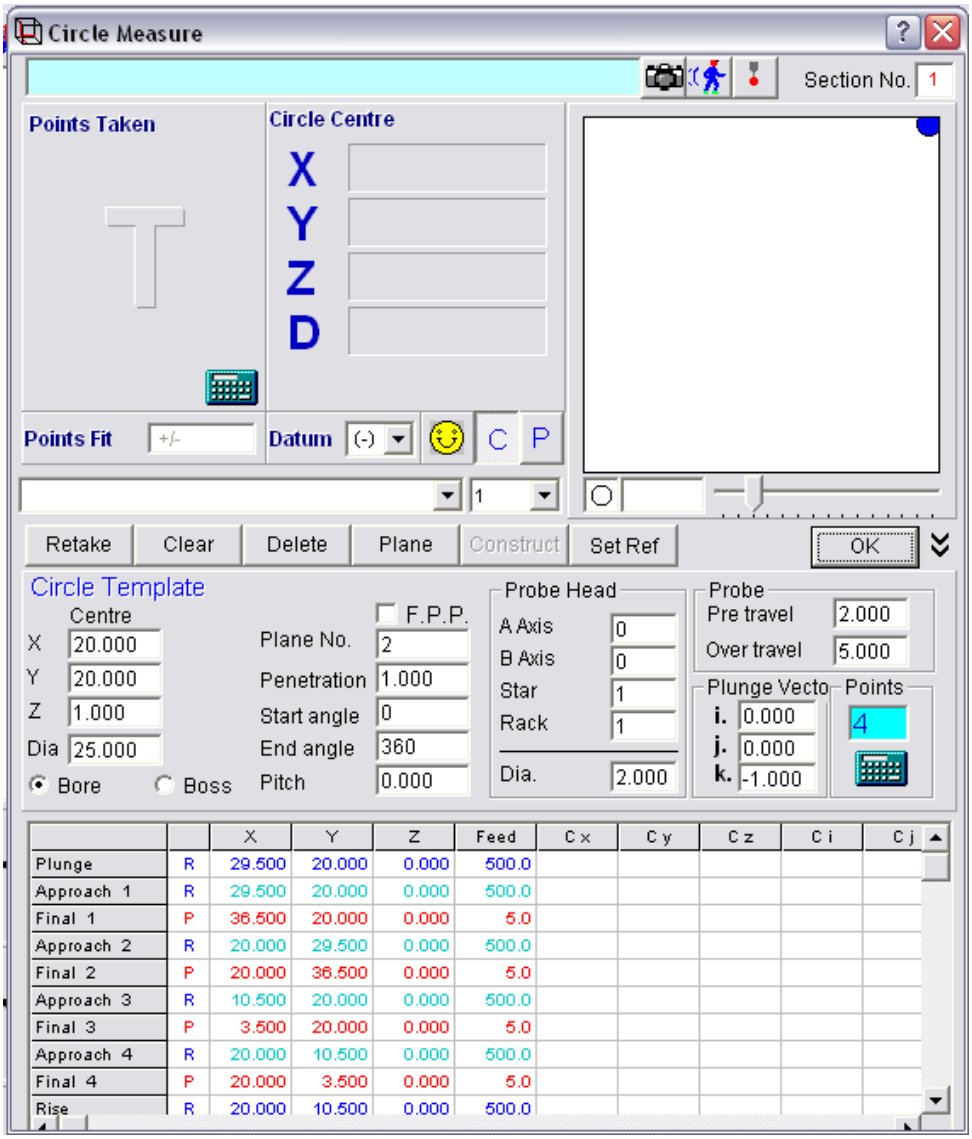


Fig. 11.1.2-2 'Circle Template from Details button'

Note that the Plane No. has now been filled in as 2. This means thats the plane the software thought was most appropriate for the circle was feature no. 2. You could change the plane no. to a more appropriate feature, if desired, and then you would have to recalculate the template by clicking on the 'Template from Details Given' button again.

You can now click on OK. The circle will appear as a grey template in the main screen.

If you call up any dimensions relative to this circle they will be shown in brackets, as at the moment the circle is just a program template, ie. It has not yet been measured.

www.EngineeringBooksPdf.com

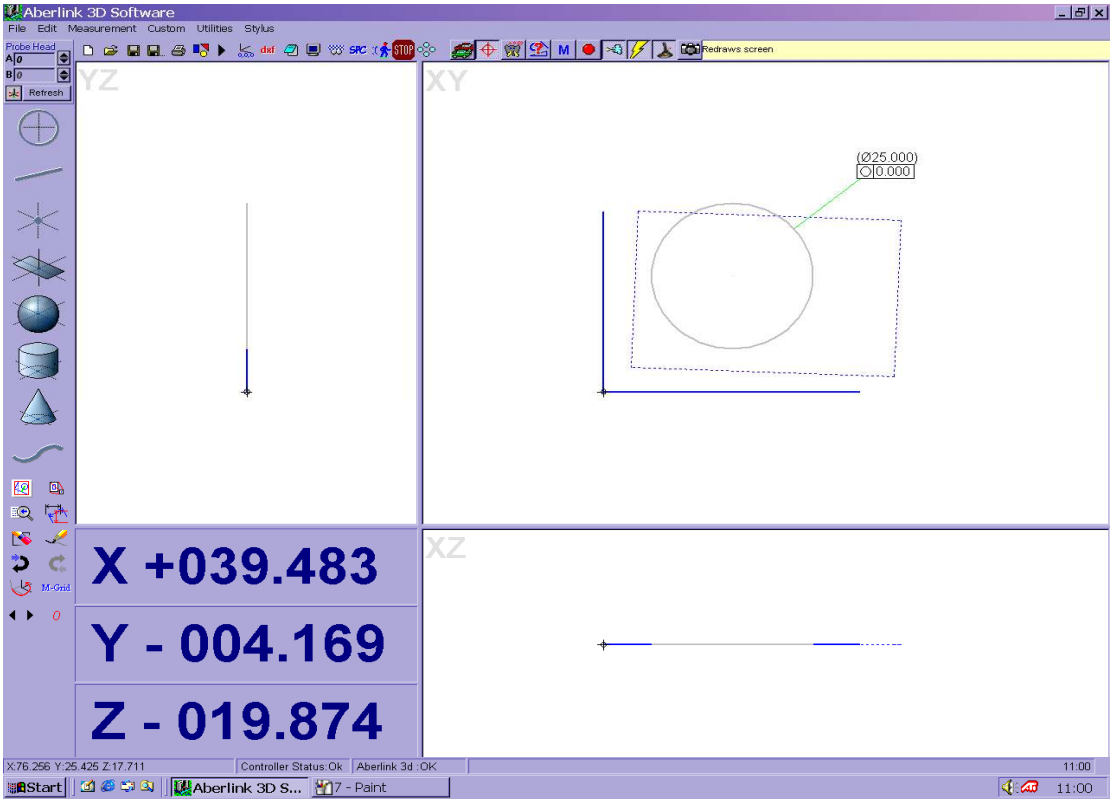


Fig.11.1.2-3

When the program has been run and the circle is measured then the dimensions will no longer be bracketed.

If you click on the 'Display Probe Contact Points' button the measurement points will be displayed as dots:

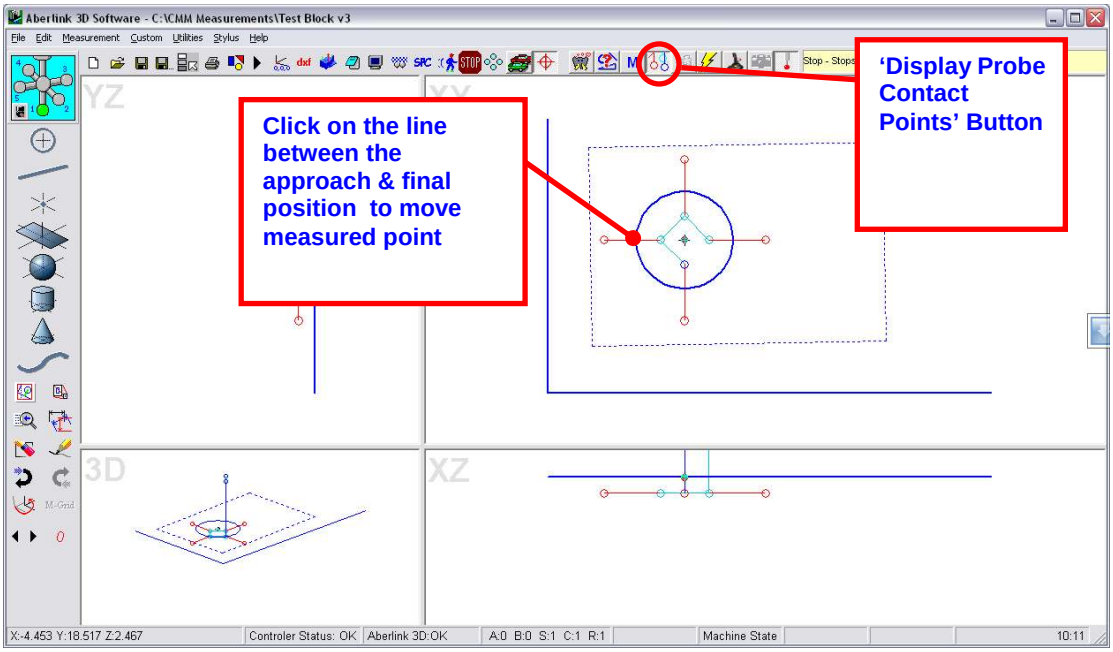


Fig. 11.1.2-4

www.EngineeringBooksPdf.com

If any of the points on the circle were in an inappropriate position (for instance they coincided with a slot or other feature) there are 2 courses of action available:

1. Move the point to a more suitable position
   With the 'Display Probe Contact Points' button pressed so that the measurement points are visible, simply click on the measurement line that connects the approach point to the final point and drag it to a convenient position on the circumference of the circle. When you release the mouse button the point will snap to the closest position on the circle from where you release it. See 38.0 Displaying probe contact points

2. Erase the point
   In this case recall the Circle Measure window and from the probe moves grid delete the 2 lines that go to make up the point. For instance if you wish to erase point number 3, click on the box that says 'Approach 3', then right click and select 'Delete Cells'. Repeat the process for the move called 'Final 3'.

11.1. 3 The 'Template from Measured Data' button

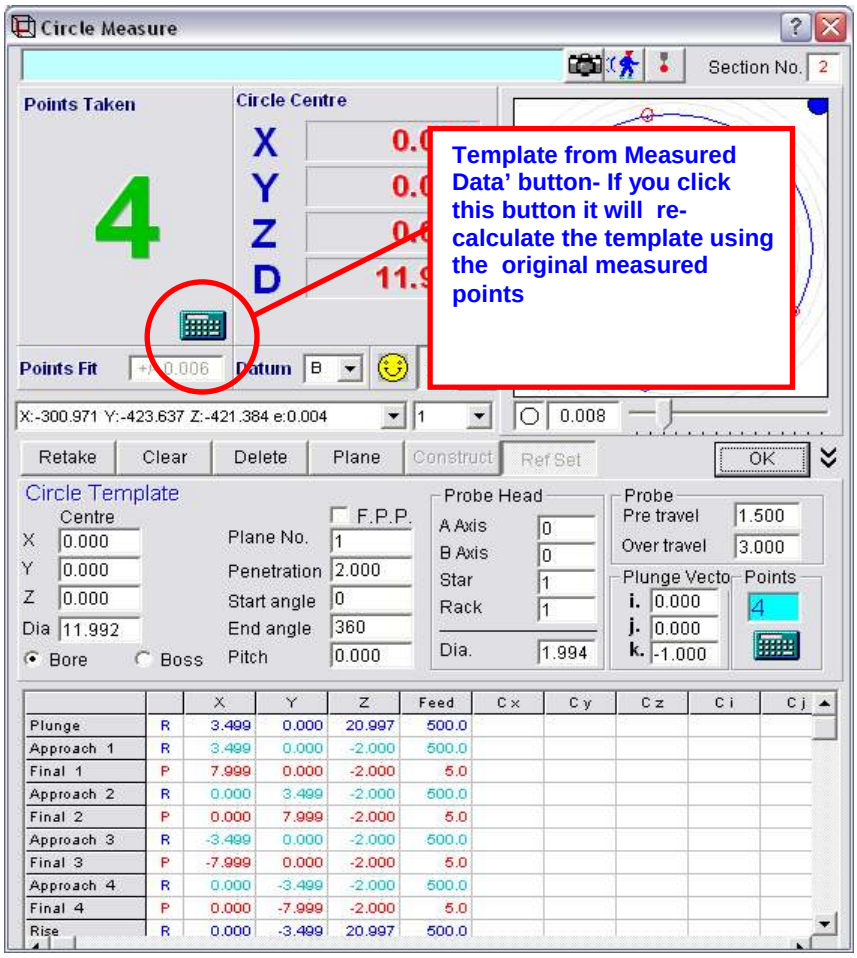


Fig 11.13-1 'Template from Measured Data' button

www.EngineeringBooksPdf.com

When you create a program unit using the 'teach and repeat' method the program template is automatically calculated from measurement points taken.

If however, you need to re-calculate the template at any time, but cannot use the template details (for instance the points cannot be equi-spaced) then it is possible as follows:

- Clear the existing points, by clicking on the 'Clear' button.
- Take some new measurement points with the probe.
- Click on the 'Template from Measured Data' button to create a new program template from the points taken.

Note when you click on the 'Template from Measured Data' button the probe moves grid is re-calculated using the points taken. These points are shown in the top left hand corner of the window.

It is bad practice to re-calculate the template for a feature that has been set as a reference once the program has been run.

11.1.4 When to OK a Circle, and when to reference.

Having measured a circle you can click on the 'OK' button to close the window or click on the the 'Set Ref ' button to make the circle a reference feature.

Setting a circle as a reference will set it's centre co-ordinates to (0,0).

If you select 'OK' the centre co-ordinates of the circle will be relative to any references previously set, or if none have been set then to the machine's own reference point.

Note – Only one circle can ever be set as a reference at any point in time. If a second circle (or other point) is set as a reference, this will override the initial reference point, and the centre co-ordinates of the second circle or point will be set to 0,0.

When you click on the 'OK' button, the measurement window will close, and the program will return to the main screen. If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position.

www.EngineeringBooksPdf.com

11.1.5 Measuring a circle in a user defined plane

If the software should have trouble in defining which plane a circle is in, or if it is in a plane which cannot be defined by the orthogonal axes of the component, then it is still possible to measure the circle by the user defining the correct plane. Having clicked on the Circle Measure button to bring up the Circle Measure window, click on the ‘Plane’ button. This will bring up a window as follows:

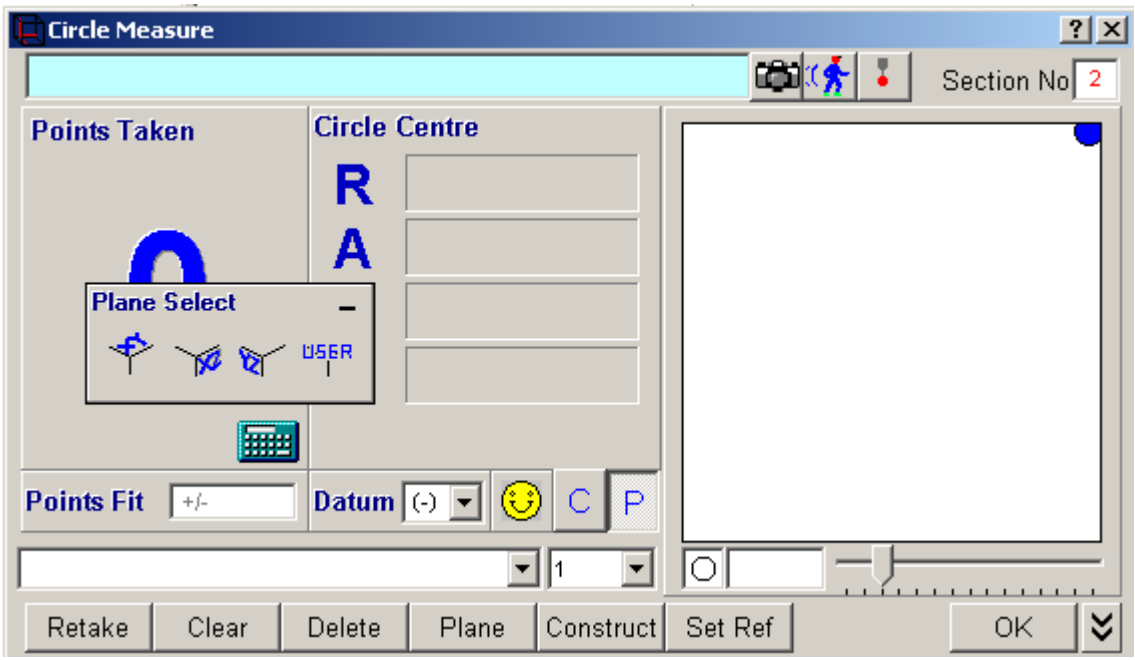


Fig. 11.1.4-1 The Circle Measure widow showing the Plane Select feature

If you are simply defining the correct orthogonal plane, just click on the appropriate XY,XZ or YZ button, and then continue with the circle measurement. If the required plane is not one of the orthogonal planes, you can still measure the circle, as long as the relevant plane has previously been measured. Now click on the ‘User’ button, and the following prompt will appear:

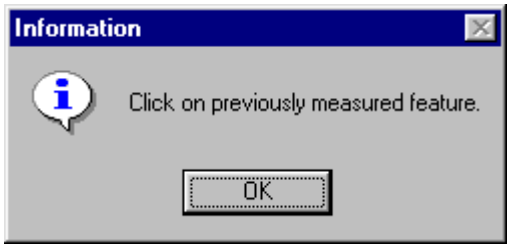


Fig. 11.1.4-1 The prompt telling you to select a previously measured plane

Now simply click on ‘OK’ and then click on the previously measured plane before continuing with the circle measurement in the usual way.

www.EngineeringBooksPdf.com

11.2 Constructing Circles

11.2.1 Constructing a Pitch Circle Diameter

Background
Pitch circle diameters can be inspected in a very similar way to measuring a circle directly. The only difference is that instead of fitting a circle through measurement points taken with the probe, you construct a circle through previously measured circles or points. The plane of measurement will be taken as that of the first point selected.

Method
To construct a pitch circle diameter, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen.

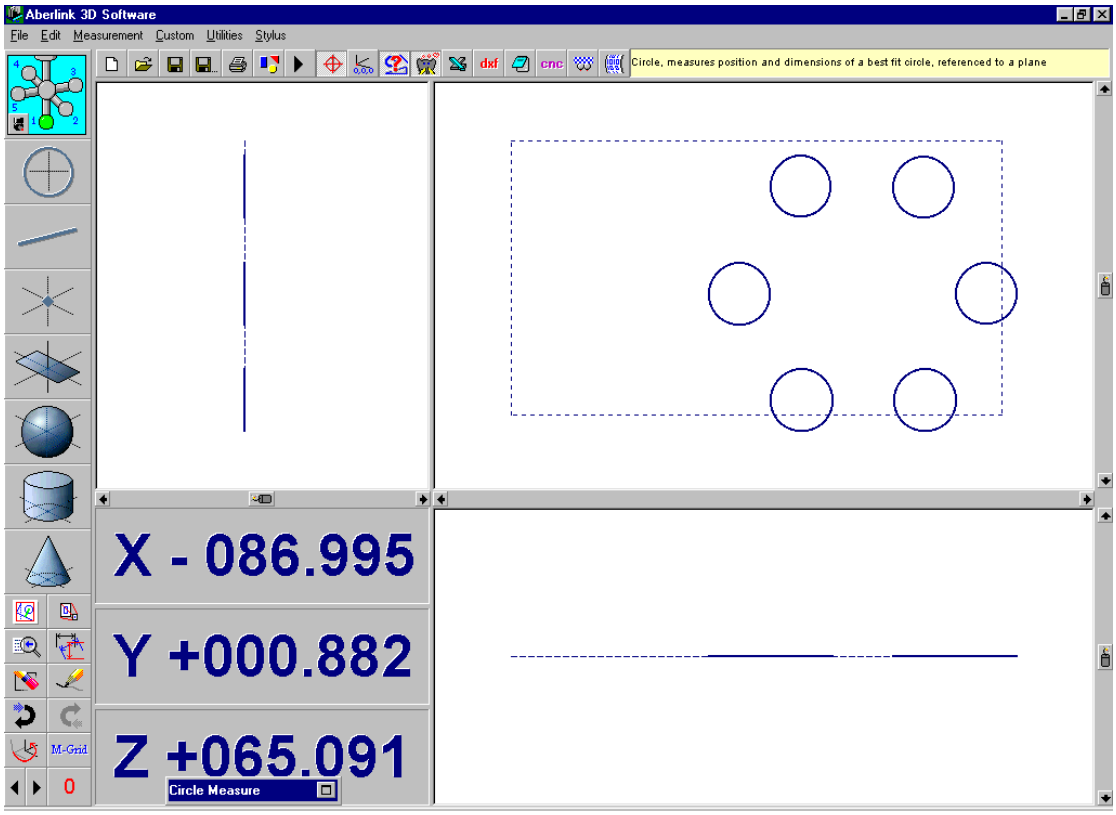


Fig. 11.2-1 The Main Screen showing a shrunken Circle Measure window

Now simply select the features that you wish to use to construct the circle by clicking on them. After you have selected the first feature a prompt will appear to confirm what is happening:

www.EngineeringBooksPdf.com

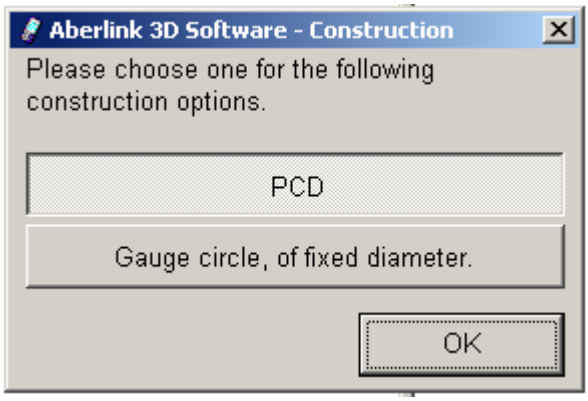


Fig. 11.2-2 The Prompt asking if you want to construct a PCD

Select the PCD option then click on Yes, and then continue to click on the remaining features that make up the PCD. When you have selected all the desired features bring the Circle measure window back by clicking on the right hand end of the shrunken box. A graphical representation of the circle will appear with values for the fit and roundness given if more than 3 points were selected.

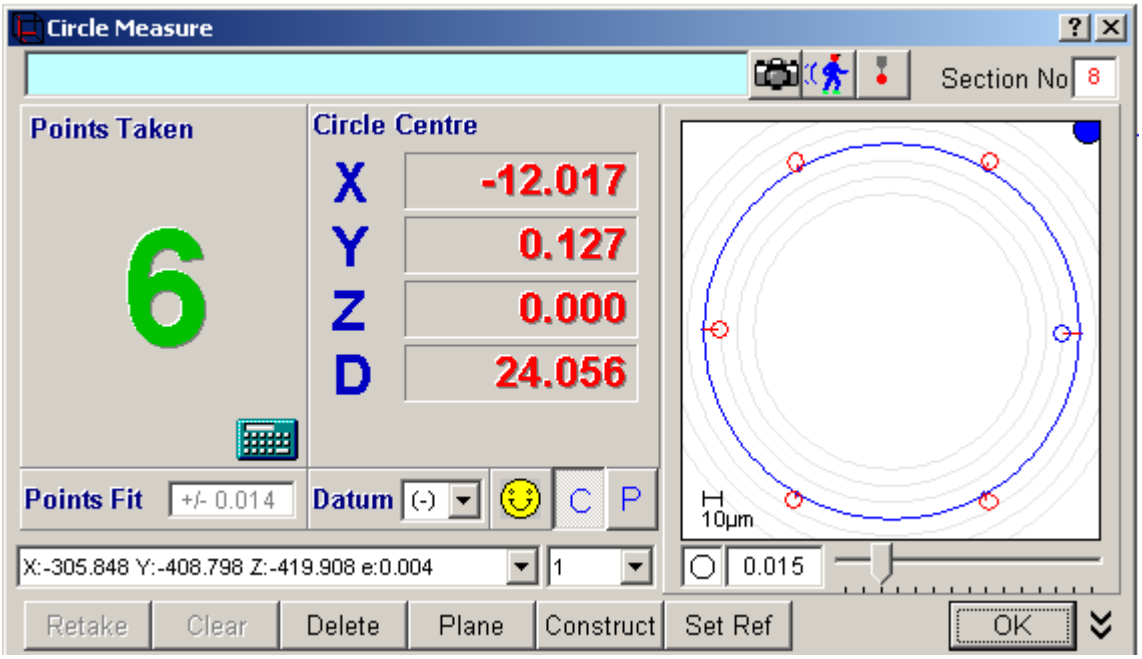


Fig. 11.2-3 A graphical representation of the constructed PCD

You close the window by click on ‘OK’ button, as for any measured circle. The screen will then return to the Main Screen, and the circle will form a part of the graphical representation, as would any measured circle.

www.EngineeringBooksPdf.com

11.2.2 Constructing a Circle at the Intersection of Two Features

Background

In addition to constructing pitch circle diameters, circles can also be constructed at the intersection of two 3-dimensional features, such as a plane and a perpendicular cylinder. In fact circles may be constructed when any of the following features overlap:

Plane/Cylinder	(if perpendicular)
Plane/Cone	(if perpendicular)
Plane/Sphere	
Cylinder/Sphere	(if on common axis)
Cone/Sphere	(if on common axis)
Sphere/Sphere	(if on common axis)
Cone/Cylinder	(if on common axis)
Cone/Cone	(if on common axis)

Method

To construct a circle at the intersection of two features, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen. Now simply select the features that you wish to use to construct the circle by clicking on them. After you have selected the first feature one of two prompts will appear, to confirm what is happening, for instance:

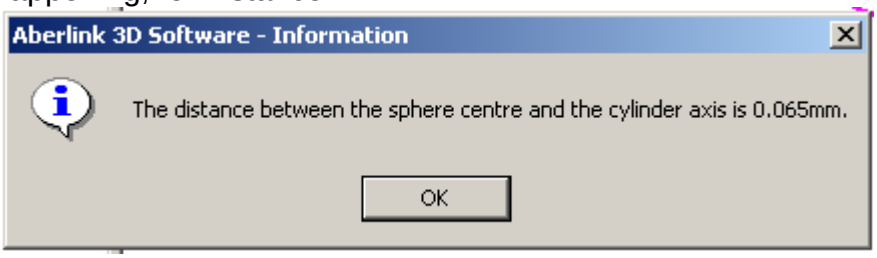


Fig. 11.2-4 Prompt for constructing a circle at an intersection

Click on OK, and then click on the other feature that creates the desired circle. If you are creating the intersection of a cylinder or cone with a plane, a different prompt will appear, telling you the angle between the cylinder or cone and the plane

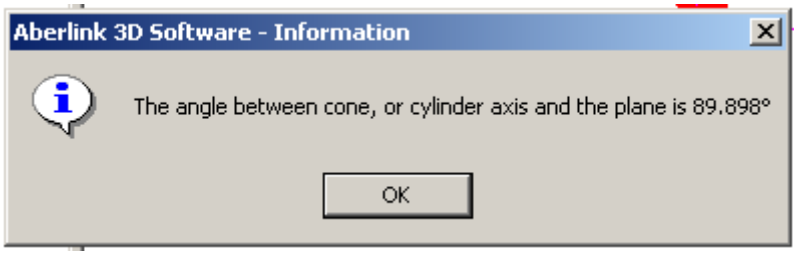


Fig. 11.2-5 Prompt stating angle between features

This is because if the features are not truly perpendicular, their intersection will not be a perfect circle, and this information allows the user to make a

www.EngineeringBooksPdf.com

judgement of the validity of the circle that they are constructing. Now click 'OK'.

The Circle Measure window will reappear. The number of points taken will be 0, and the graphics part of the window will be blank:

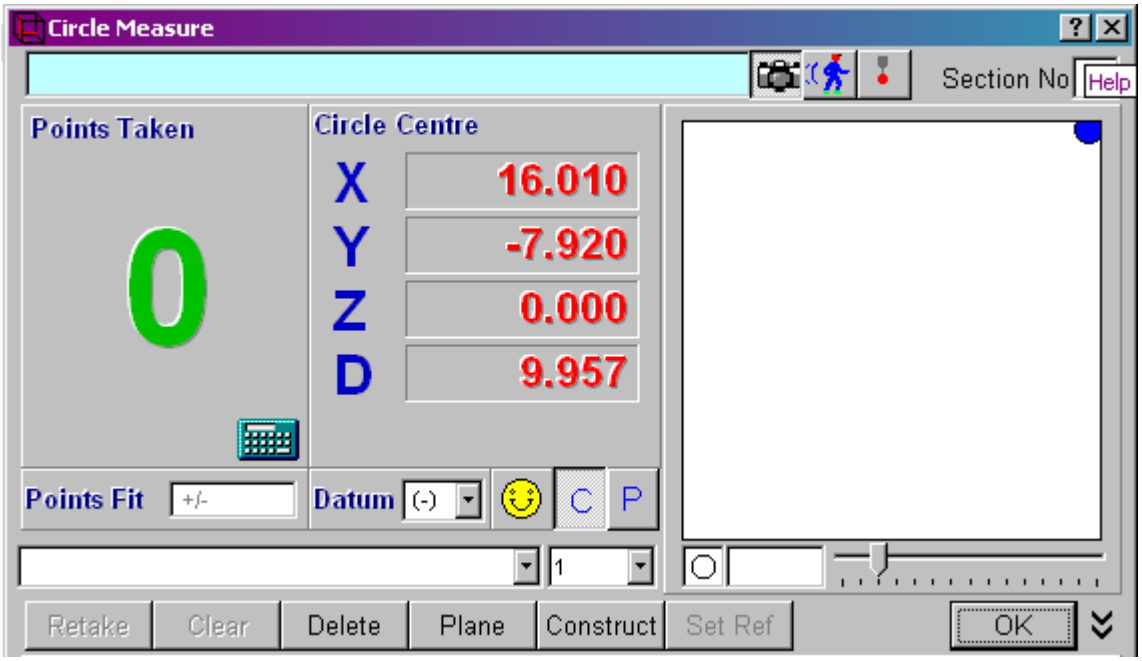
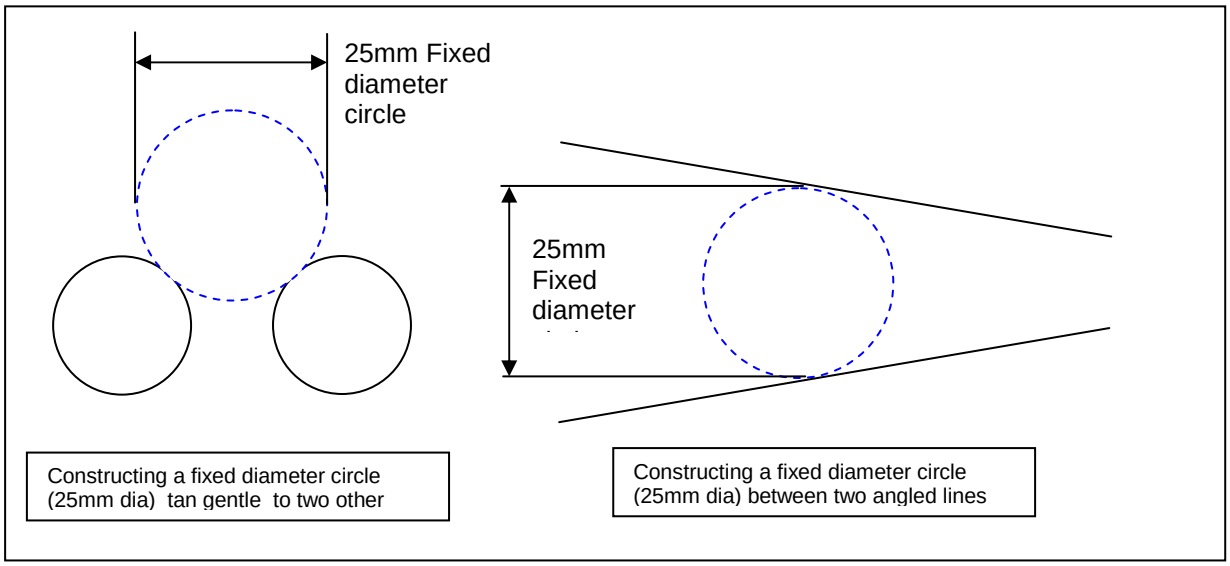


Fig. 11.2-6 No representation is given when constructing a circle at an intersection

You may now click on 'OK' or 'Set Ref' as appropriate, as for any measured circle. The screen will then return to the Main Screen, and the circle will form a part of the graphical representation.

www.EngineeringBooksPdf.com

11.2.3 Constructing of gauge circle of a fixed diameter between two points or lines



Background

It is now possible to construct a fixed diameter circle tan gentle to two measured circles or between two angled lines.

Method

To construct a fixed diameter circle between two other features, click on the Circle Measure button, from the main screen. This will bring up the Circle Measure window. Now click on the 'Construct' box. The Circle measure window will now shrink to a small box at the bottom of the screen.

Now simply select the first feature that you wish to use to construct the gauge circle through by clicking on it. After you have selected the first feature a prompt will appear, to confirm what is happening.

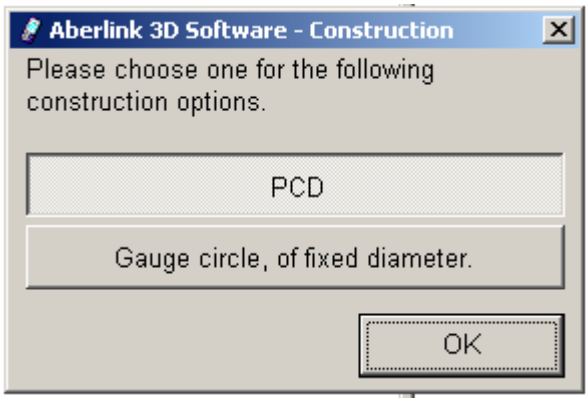


Fig. 11.2-7 The Prompt asking if you want to construct a Gauge circle

Select the 'Gauge circle, of fixed diameter' option then click on Yes a second prompt window will appear asking for the diameter of the gauge circle

www.EngineeringBooksPdf.com

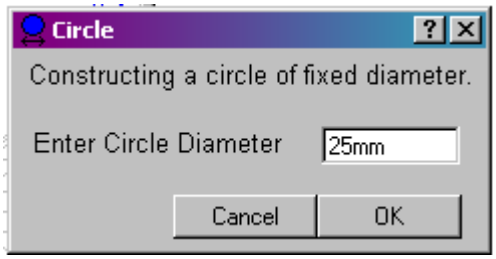


Fig. 11.2-8 The Prompt asking for the diameter of the Gauge circle

Enter the diameter of the gauge circle to be fitted between the two features, Click on ‘yes’
Finally click on the second feature bring the Circle measure window back by clicking on the right hand end of the shrunken box. Then click ‘OK’ The screen will then return to the Main Screen, and the circle will form a part of the graphical representation.

www.EngineeringBooksPdf.com

12.0 THE POINT UNIT

Points are an infinitesimally small position in space, which will have an X,Y and Z co-ordinate, and can be measured directly using the probe.

Points can also be constructed at the intersections of various features, or at the mid-point of a line.

Points may also be constructed at co-ordinates that can be entered via the keyboard.

12.1 Measuring Points Directly

Background

Measuring a point directly will give a single point co-ordinate on the surface of the component. As long as the direction of the measurement point has been defined using the direction button, the software will apply the stylus radius offset, by looking at the direction of motion of the probe when the measurement point is taken.

As the probe can strike the component only in one direction, only one of the X,Y,Z co-ordinates will have any real meaning. This should be noted when bringing up dimensions to a measured point.

Setting a measured point as a reference can define the position of a plane along the relevant axis of the point, ie. It will define 1 degree of freedom for the component, see chapter 5 – Aligning the Axes of a Component.

Method

To measure a point, click on the Point Measure button, from the main screen:

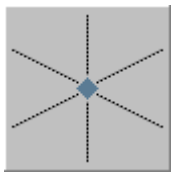


Fig. 12.1-1 The Point Measure button

This will bring up the Point Measure window:

www.EngineeringBooksPdf.com

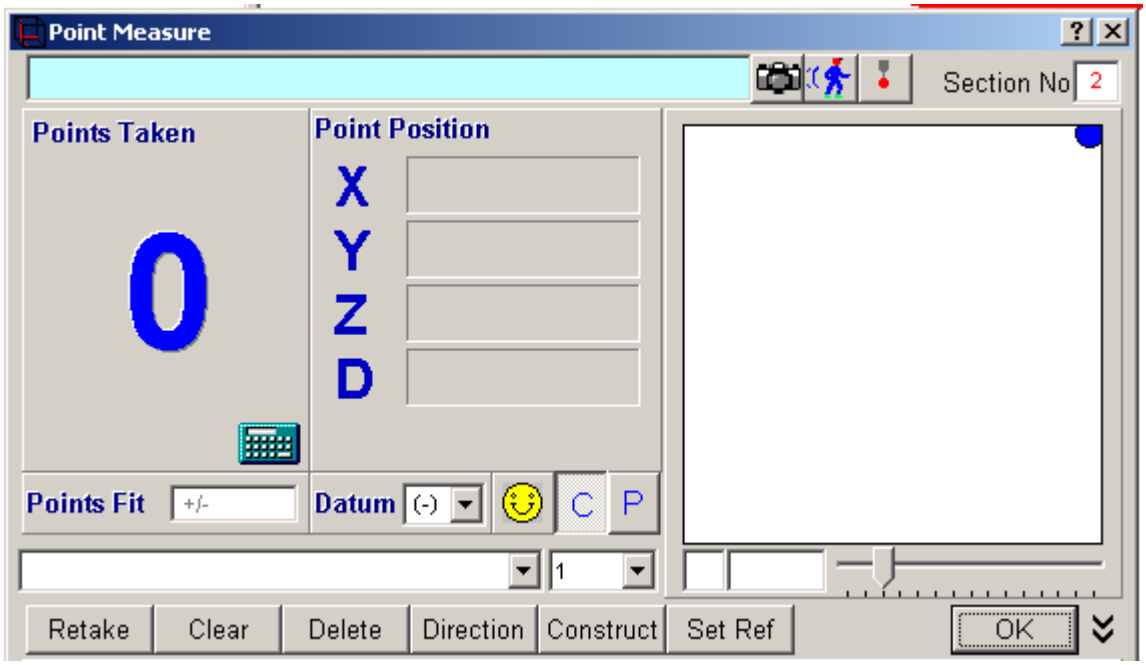


Fig. 12.1-2 The Point Measure button

Click on the 'Direction 'button and the following prompt will appear:

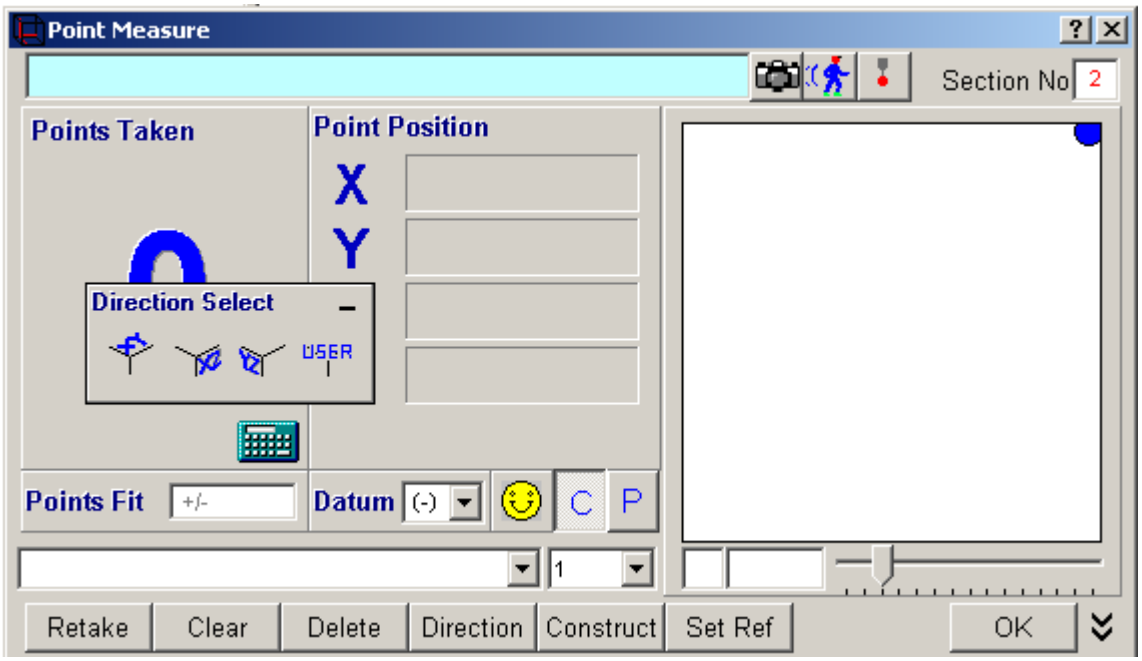


Fig. 12.1-3 The select direction prompt

You now need to pick one of four options depending on the direction you are measuring in:

www.EngineeringBooksPdf.com

1. If you are moving the probe along the Z axis to measure a point you need to select the ‘XY’ option

2. If you are moving the probe along the X axis to measure a point you need to select the ‘YZ’ option

3. If you are moving the probe along the axis to measure a point you need to select the ‘XZ’ option

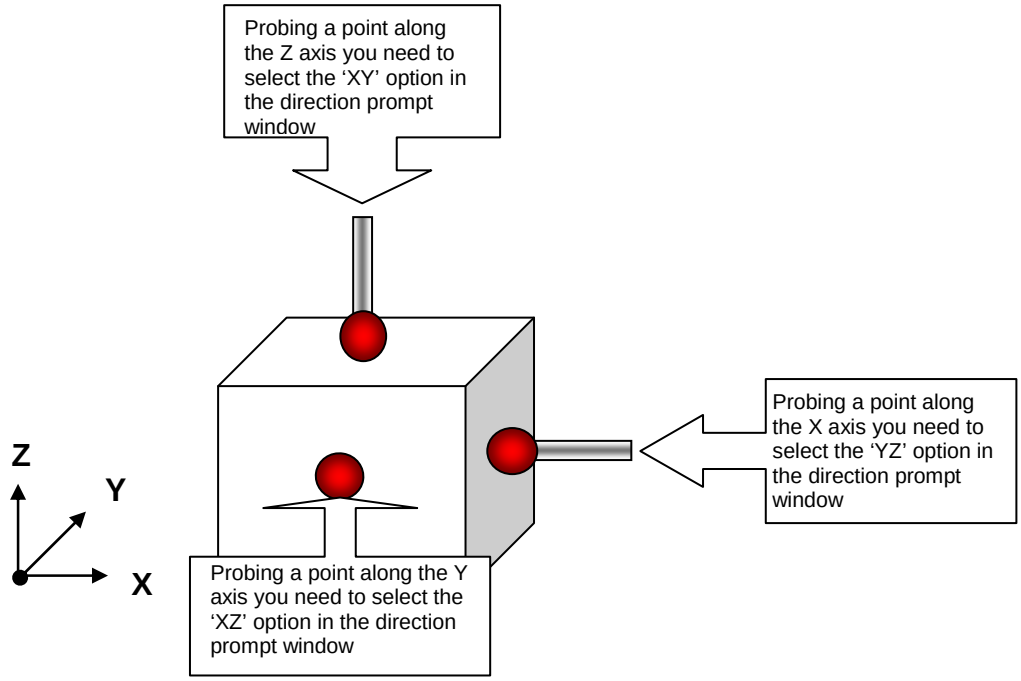


Fig. 12.1-4 Diagram showing you how to select a direction when measuring a point

Now take a measurement point on the component at the position that you wish to measure.

4. If you are measuring a point on a surface which is offset to the machine co-ordinate system you need to select the ‘user’ option from the direction window. When you click on the ‘user’ button, the following prompt will appear:

www.EngineeringBooksPdf.com

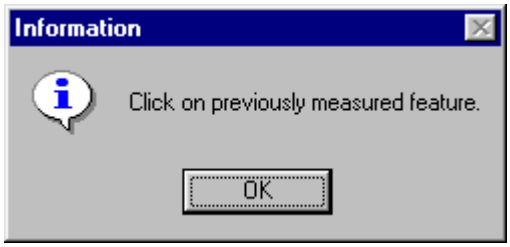


Fig. 12.1-5 The prompt telling you to select a previously measured feature

Now simply click on 'OK' and then L/H click on a previously measured feature before continuing with the point measurement in the usual way.

After the correct direction for the point has been defined the software looks at the route that the probe was moving along when it took the point and then it applies the correct stylus radius offset to that point.
This means that the measured point will actually lie on the surface of the part being measured and not at the centre of the stylus ball. If you have been successful the X,Y and Z co-ordinates will appear in red.

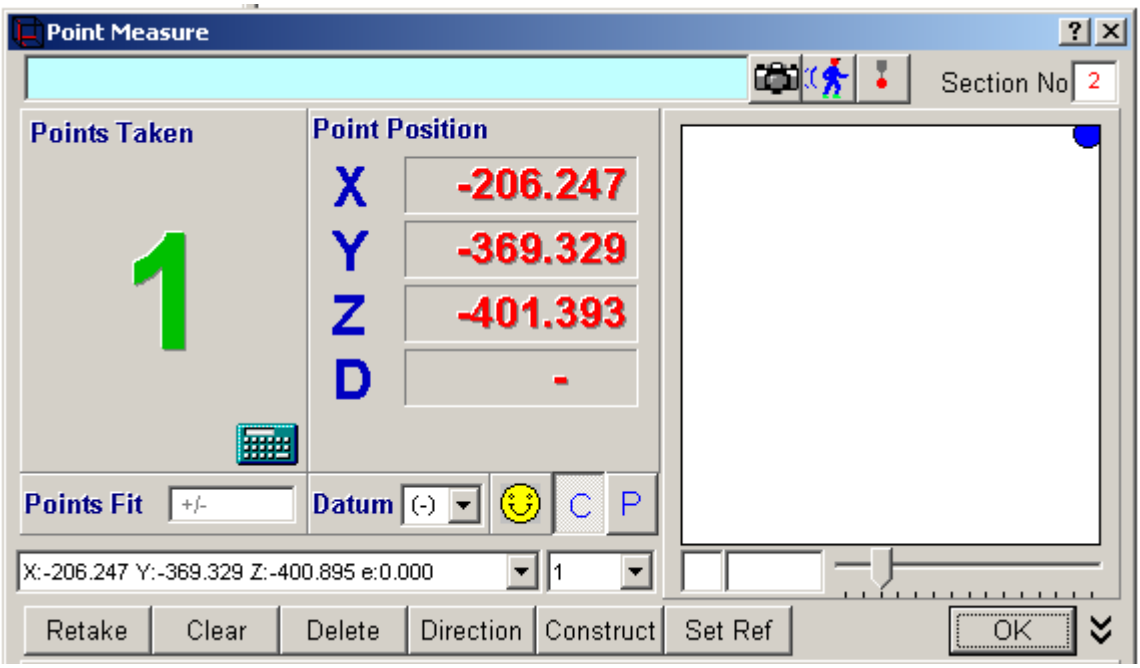


Fig. 12.1-5 Showing a point in a plane that has been defined

As a point measurement is a single point, the smiley face warnings are irrelevant and the Point Fit box and graphics part of the window will always remain blank.

If you have accidentally probed on the wrong place on the workpiece, or are unhappy with the point, simply take the point again. As the point measurement function is only ever a single point, this will override the initial point taken.

www.EngineeringBooksPdf.com

12.1.1 Displaying the centre position in polar co-ordinates

By default the centre co-ordinates of points are displayed as X,Y and Z values (carthesian co-ordinates). In order to display them in polar co-ordinate form, simply click on the ‘P’ button, situated below the ‘Point Fit’ box.

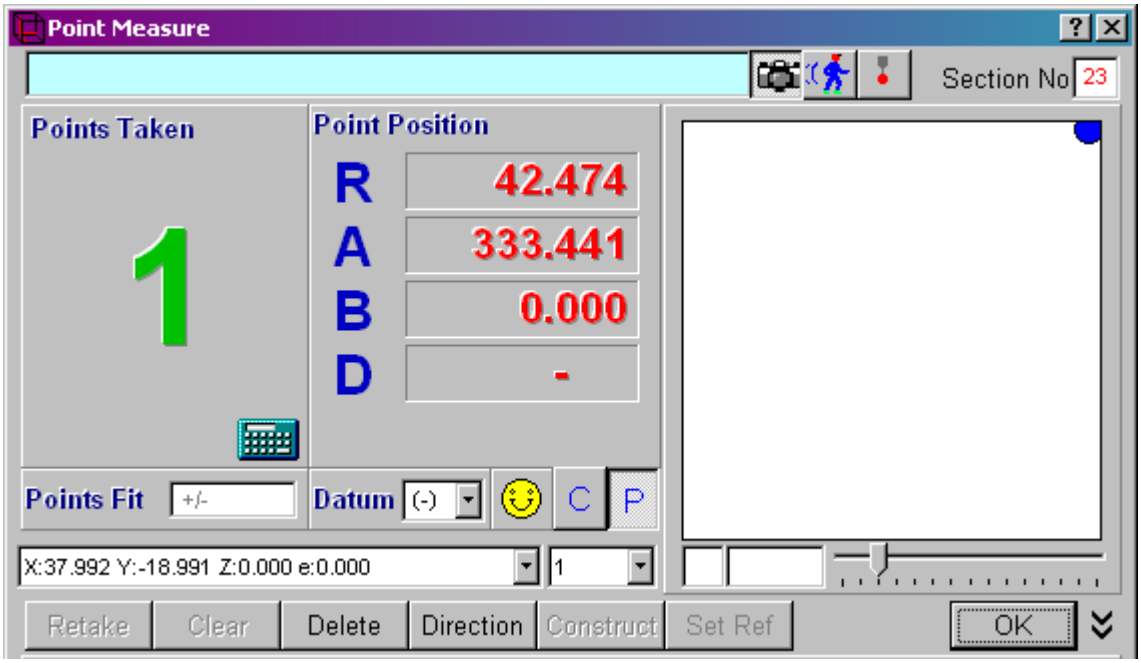


Fig. 12.1-6 Showing a point displayed in polar co-ordinates

Now instead of displaying X,Y and Z values for the point, the screen will now display R, A and B values. This is now in polar co-ordinate form, with the R representing the distance from the X,Y origin, A being the angle in XY, and B being the elevation angle from the XY plane.

Note – that for these values to have any real meaning, a correct X,Y origin must have previously been set.

To revert to carthesian co-ordinates simply click on the ‘C’ button.

12.1.2 Off Line Programming

Instead of directly measuring a point using the probe it is possible to enter the co-ordinates for the point in order to create a measurement program for it.

For instance if we want to measure the Z height on a component at the position X=5, Y=5 we can produce an off-line program for the point as follows:

Click on the Point Measure button, from the main screen to bring up the Point Measure window. Now instead of taking the point with the probe we are going to type in the template details directly:

www.EngineeringBooksPdf.com

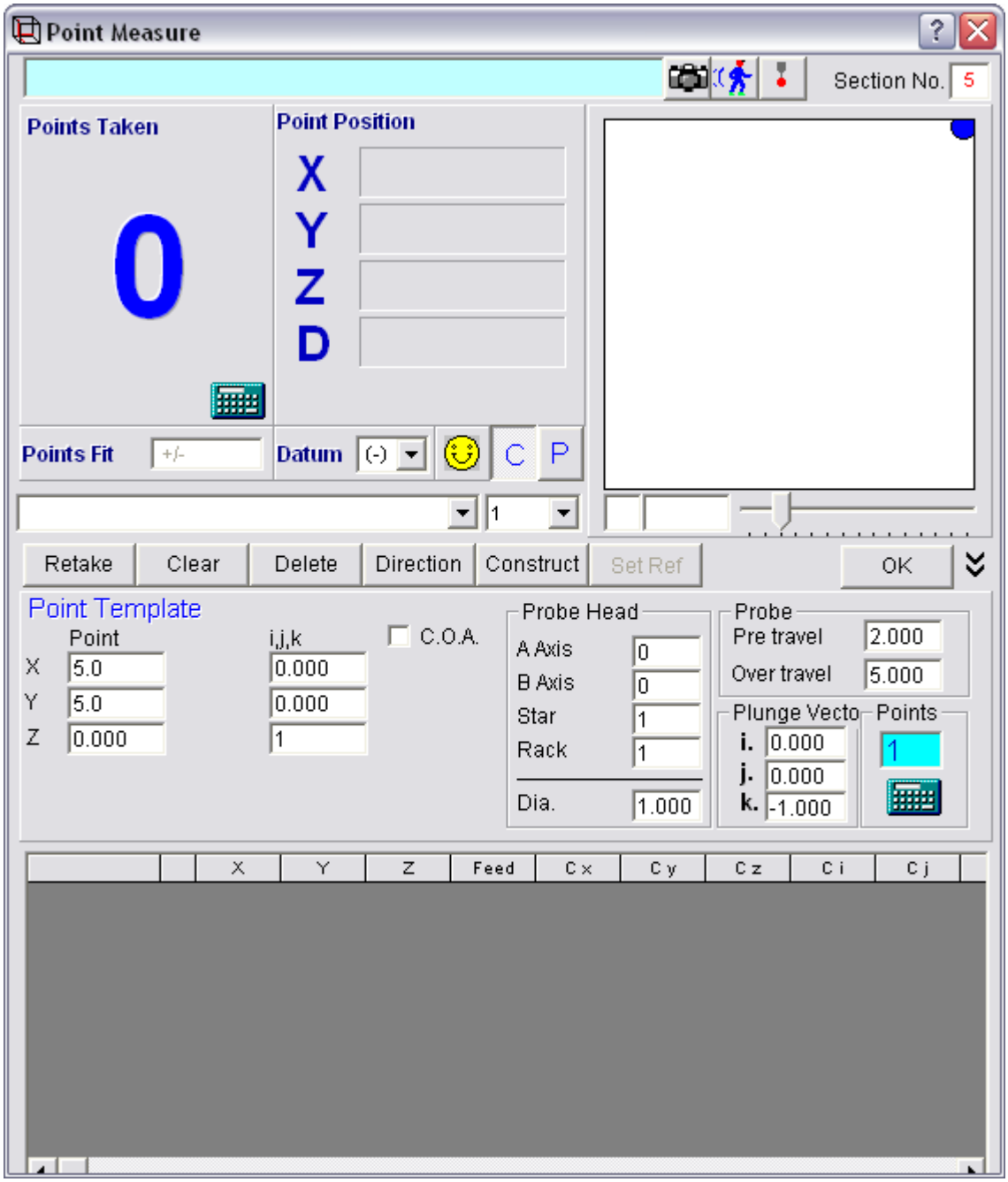


Fig.12-7

A point is defined by entering the X, Y, Z position of the point and the direction of approach for the point. In this instance if we assume that we are measuring a point on the reference plane then the position of the point will be X=5, Y=5, Z=0. The approach for the point wants to be in a straight down direction so the Approach vector should be 0,0,1 (plunge vector will be 0,0,-1). Click on the 'Template from details given' button and the probe moves will be calculated:

www.EngineeringBooksPdf.com

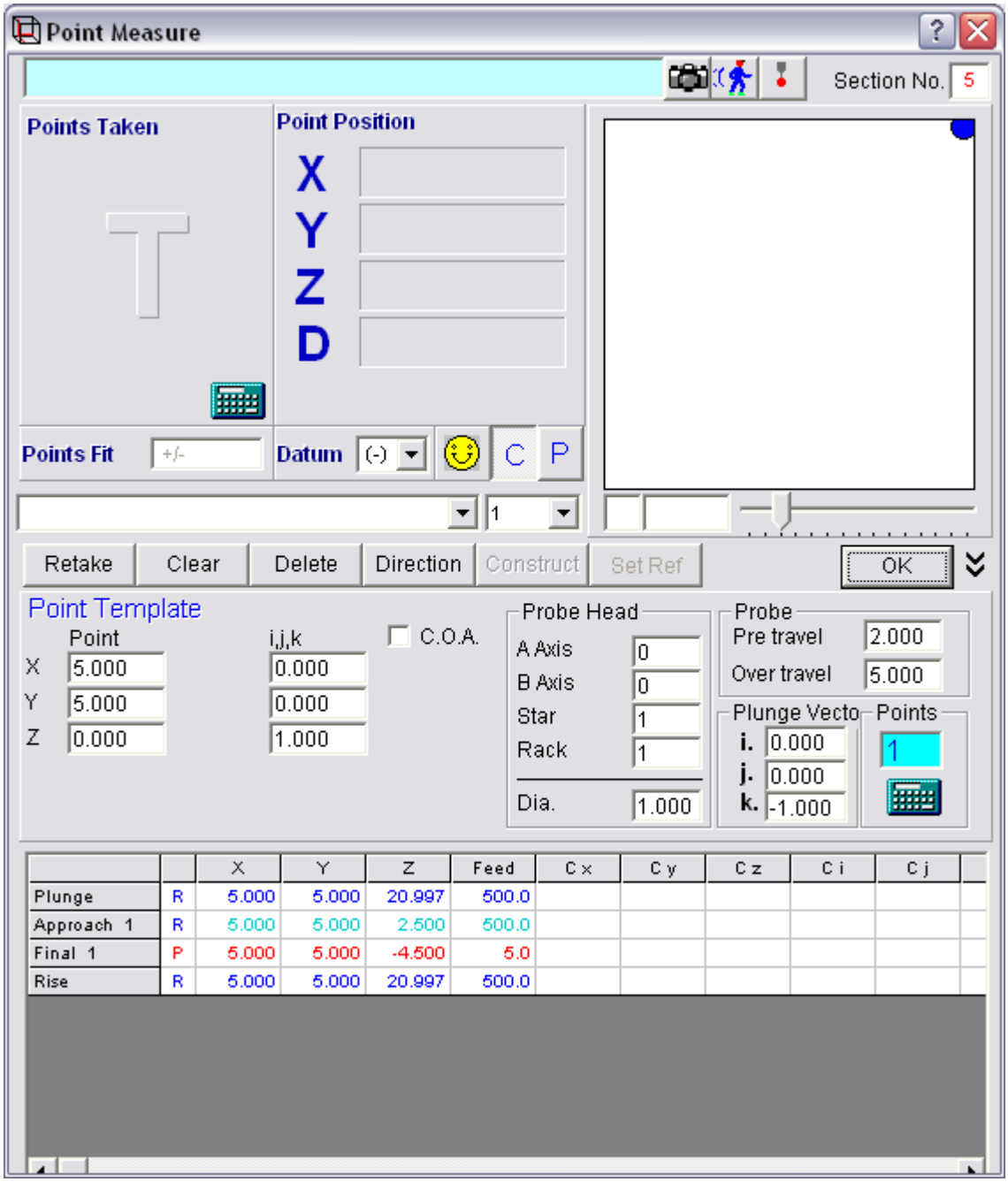
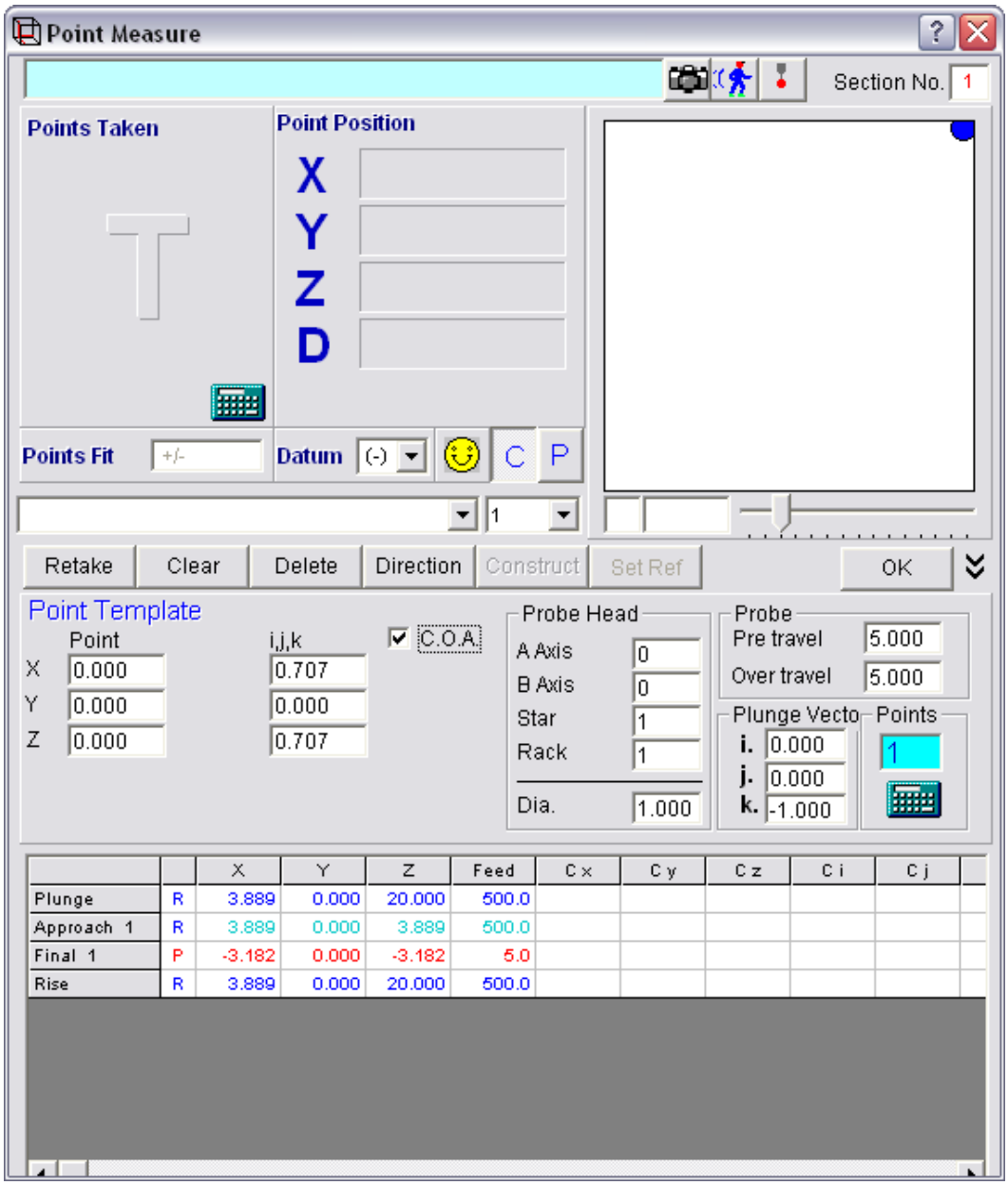


Fig.12-8

www.EngineeringBooksPdf.com

12.1.3 Measuring with Compensated Approach

The I,j,k values shown in the Point Template window gives the direction the probe moves along when measuring a Point.



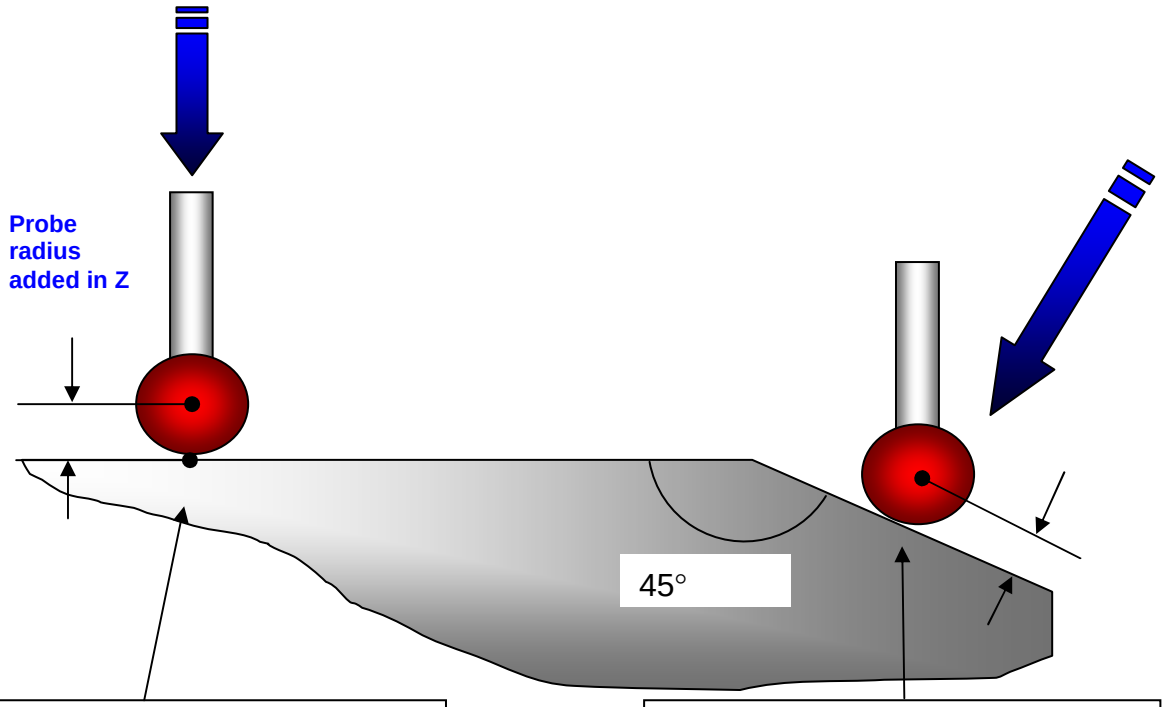
The I,j,k values (vectors) are displayed as tangents of an angle, so it is possible to work out the angle of the probe path the probe moves along by converting the tangents back in to angles, (0.707 = 45°) or if you wish the probe to move along known angle when measuring you need to convert the angle back into a tangent and then re-enter that value into the correct I,j,k box.

www.EngineeringBooksPdf.com

The Compensated Approach tick box

Normally then you measure a point the software compensate for the probes radius in the direction of the reference plane (X,Y) you selected using the 'direction button' but if you tick the compensated approach box (C.O.A) the software will compensate for the probes diameter along the approach (I,j,k) angle.

This is used on curved surfaces were you want a true surface point.



When you measure a point the software only knows were the centre of the probe is to get a true surface point the software must add the radius of the probe in the direction you picked with the 'Defines Plane' in the Point measure window.
***Example* If you measured a point in the X-Y plane the software will add the radius in Z (I,j,k value =0,0,-1)**

When you measure a point with the compensated approach box ticked the software adds the radius of the probe in the direction set out in the I,j,k boxes in the Point measure window.
***Example* If you measured a point on a 45° angle the software will add the radius in direction of the I,j,k values 0.00,0.707,0.707 or 45°**

www.EngineeringBooksPdf.com

12.1.4 When to OK a Point, and when to reference it.

Having measured a point you can either click on the ‘OK’ button or the ‘Set Ref ‘ button and then the ‘OK’ button.

If you select ‘OK’, the co-ordinates of the point will be relative to any references previously set, or if none have been set then to the machine’s own reference point.

Setting a point as a reference will perform different functions depending on whether the point is constructed or measured. If a point has been constructed, then setting it as a reference will set it’s co-ordinates to (0,0,0).

If however, it has been measured, then it can only be projected into one defined plane, and therefore only one of its X,Y,Z co-ordinates will have any real meaning. In this case setting the point as a reference will set only the meaningful co-ordinate to zero. In this way a point can be used to define the position of the plane in which the point was taken, as long as it has not previously been defined, ie. It can define 1 degree of freedom for the component, see chapter 5 – Aligning the Axes of a Component.

Note – Only one point can ever be set as a reference at any time. If a second point is set as a reference, this will override the initial reference point.

When you click on either ‘OK’ , the measurement window will close, and the program will return to the main screen. If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position. See chapter 14 - Manipulating the Graphics.

12.2 Constructing Points

As well as measuring points directly, they can also be constructed either by typing the point co-ordinates via the keyboard, or alternatively by constructing it at the intersection of features which have previously been measured, or constructed themselves.

Points can be constructed at the intersection of many features, such as between 2 lines, lines and circles, or at the intersection of lines (including cylinders and cones) with planes, etc. Some of these features will produce a single intersection point, such as 2 lines, and some will have 2 or more intersection points, such as an overlapping line and circle.

One other use for points is that they can be constructed at the mid-point or end points of a line.

12.2.1 Entering the Co-ordinates for the Point

www.EngineeringBooksPdf.com

Method

To construct a point by entering the co-ordinates for the point via the keyboard, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. A prompt will now appear on the screen asking if you would like to enter the co-ordinates of the point.

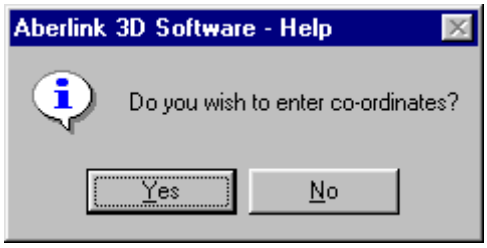


Fig.12.2-1 The prompt asking if you wish to enter the co-ordinates for the point

If you click on yes, another window will appear, allowing you to enter the X,Y and Z (carthesian) co-ordinates or the R, A and B (Polar) values of the point with the 'R 'representing the distance from the X,Y origin, 'A' being the angle in XY, and 'B' being the elevation angle from the XY plane via the keyboard.

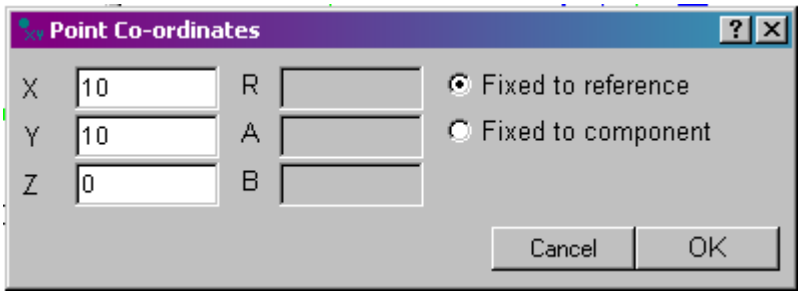


Fig. 12.2-2 The widow allowing you to enter the co-ordinates for the point

You have the choice of linking the constructed point to the reference system or to the component.

- If you link the point to the component and then rotate or translate that component the point will rotate and move with that component.
- If you link the point to original reference system it will say fixed to that original reference system even if you rotate or translate the component in the software.

Having entered your desired co-ordinates click 'OK' and the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates that you entered will be displayed in red.

www.EngineeringBooksPdf.com

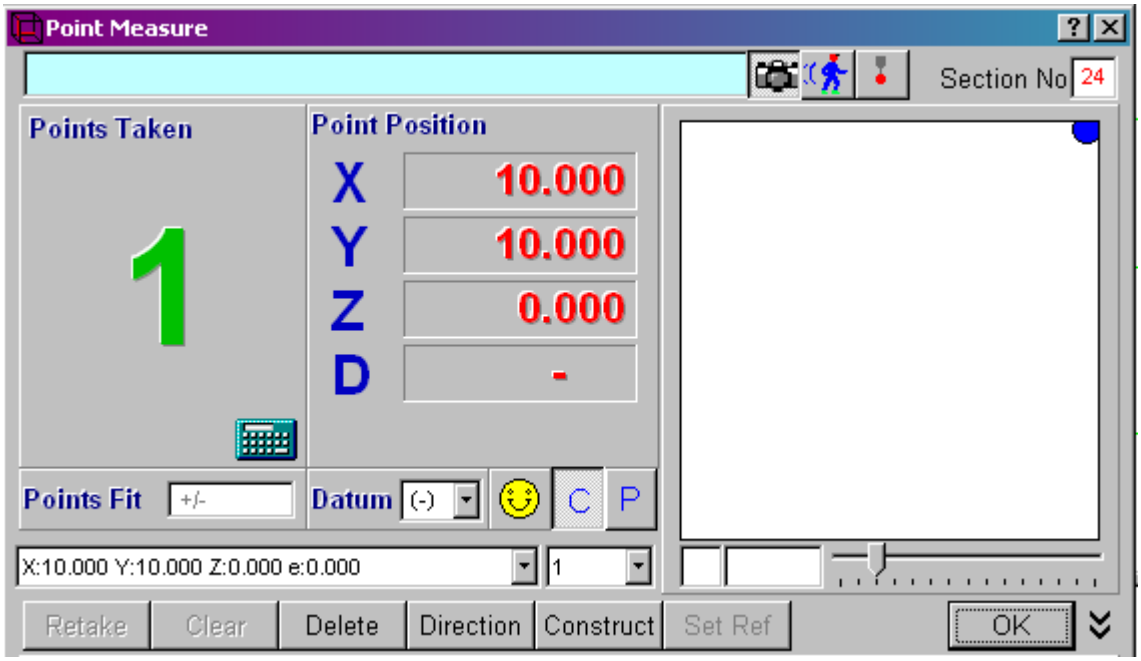


Fig 12.2-3 The Point Measure window displaying the co-ordinates of the point entered

You may now click on 'OK'
The screen will now return to the Main Screen, and the point will form a part of the graphical representation.

12.2.2 Constructing points between features with a single intersection

Points of intersection can be constructed not only at the intersection of features which touch, but can also be used to produce the closest point to both features, when they do not touch.

To construct such a point, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. The Point Measure window will now shrink to a small box at the bottom of the screen.

www.EngineeringBooksPdf.com

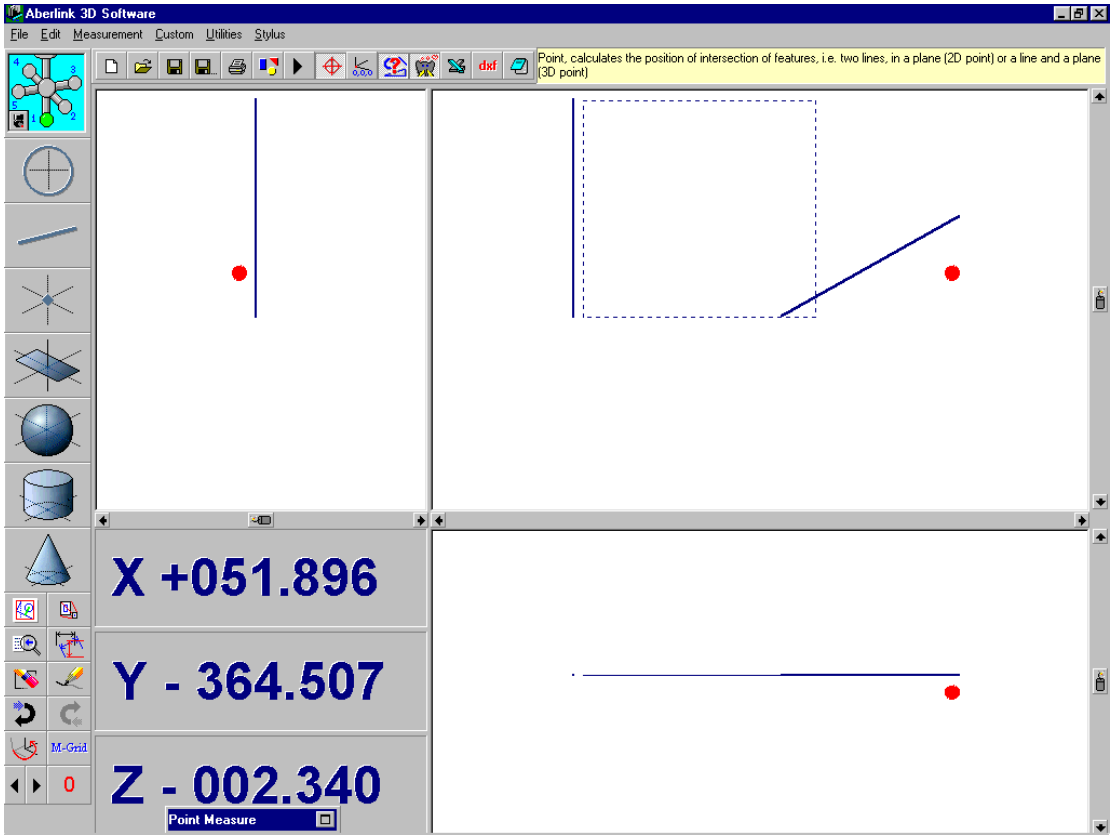


Fig. 12.2-4 The Main Screen showing a shrunken Point Measure window

Now simply select one the features, whose intersections create the desired point you want, by clicking on it.

After you have click on the second feature the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates of the constructed point will be displayed in red.

www.EngineeringBooksPdf.com

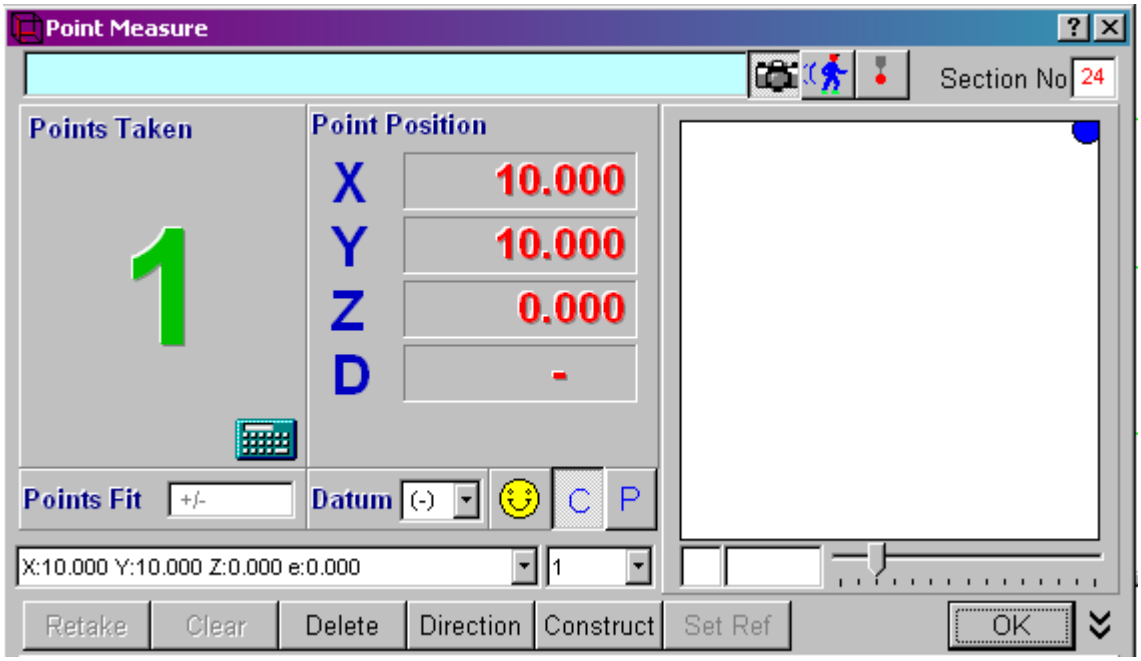


Fig. 12.2-6 The Point Measure window for a point constructed at an intersection

You may now click on ‘OK’ or ‘Set Ref’ plus ‘OK’ as appropriate, as for any point. The screen will now return to the Main Screen, and the point will form a part of the graphical representation.

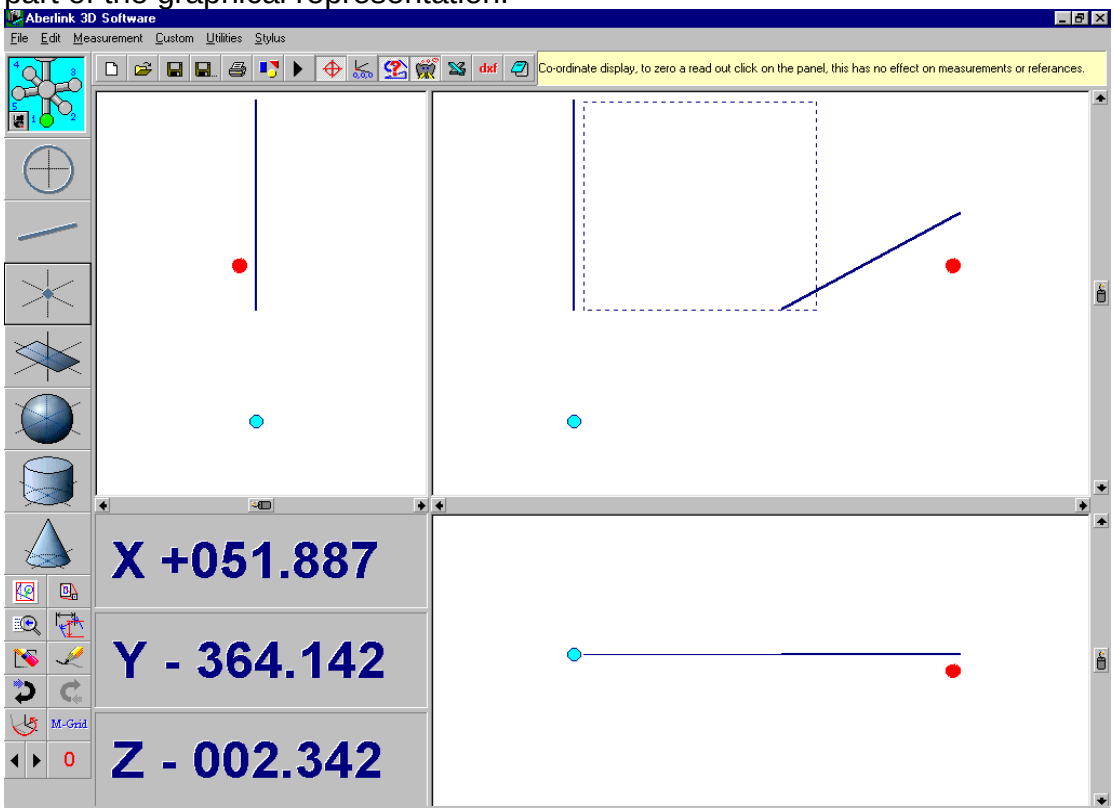


Fig. 12.2-7 The Main Screen showing a point constructed at the intersection of 2 lines

www.EngineeringBooksPdf.com

12.2.3 Constructing Points between features when there is more than one intersection

There are a few situations where the intersection of features can construct more than one point, such as two overlapping circles have 2 points of intersection. In this instance the intersection point closest to the cursor when the last feature is selected will be constructed.

Note – Using the feature select function will allow you to always select a position close to the required intersection

For example when constructing an intersection point between two overlapping circles, if you select the second circle as shown, then the following intersection point will be created.

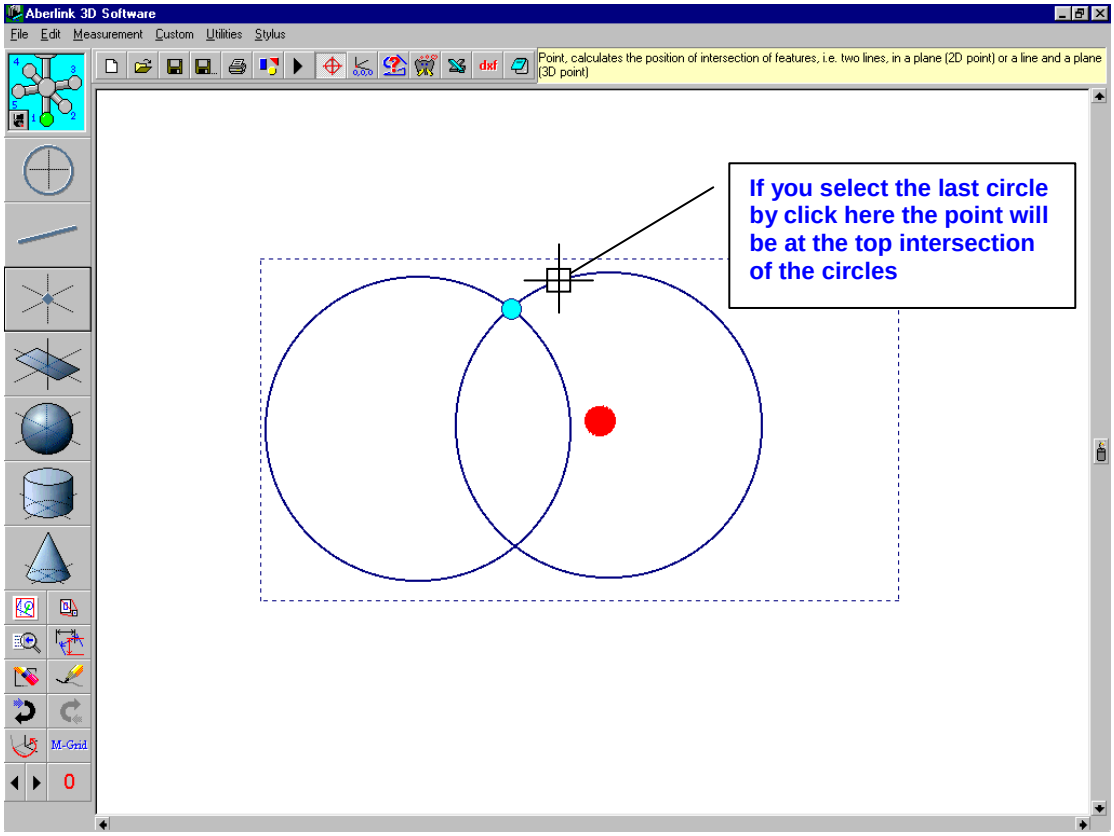


Fig. 12.2-8 Showing a constructed intersection point between 2 circles

Now to select the other intersection point simply repeat the exercise, but this time select the second circle at the position shown below:

www.EngineeringBooksPdf.com

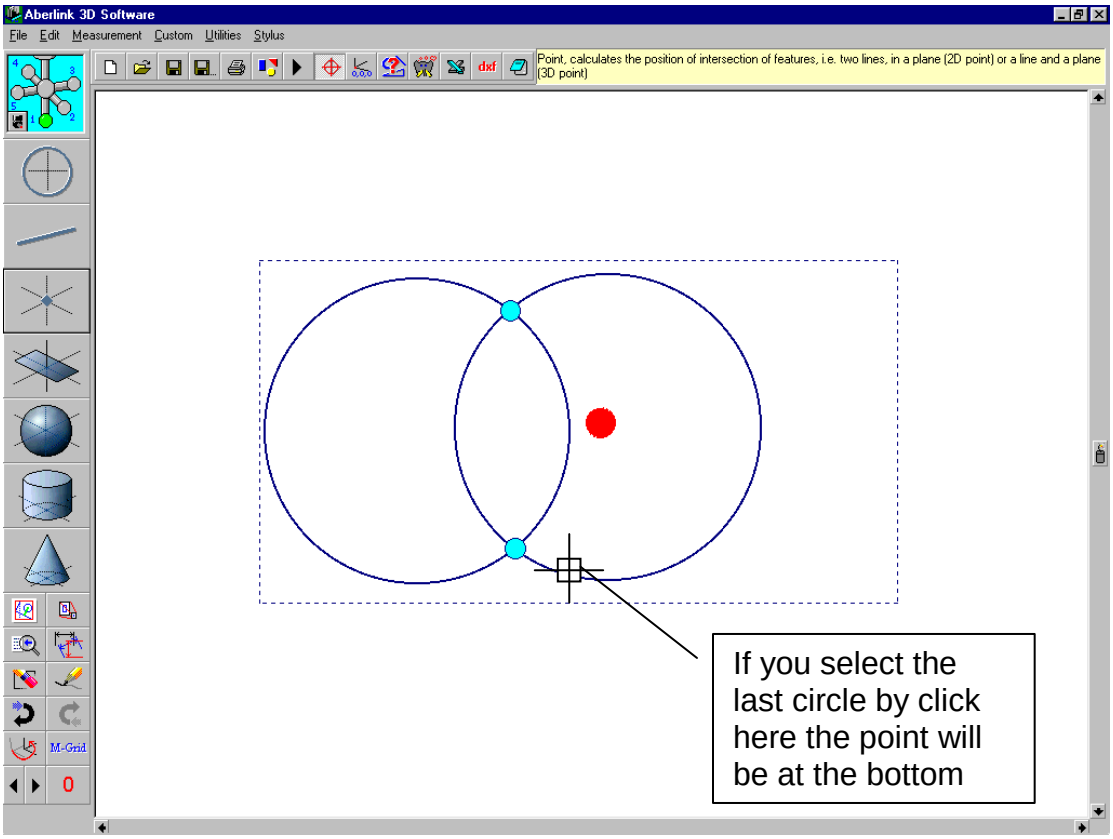


Fig. 12.2-9 Showing both constructed intersection points between 2 circles

12.2.4 Constructing Points between two cylinders

A special case is the intersection of two cylinders. In this instance there are 5 possible intersection points, namely the intersection of the two centre lines, and then 2 intersections for each centre line with the edges of the other cylinder.

Now when you are constructing these intersections they will appear in the following manner

The first constructed point will be the intersection of the 2 centre lines (or closest point if they do not exactly meet), irrespective of the order in which the cylinders were selected.

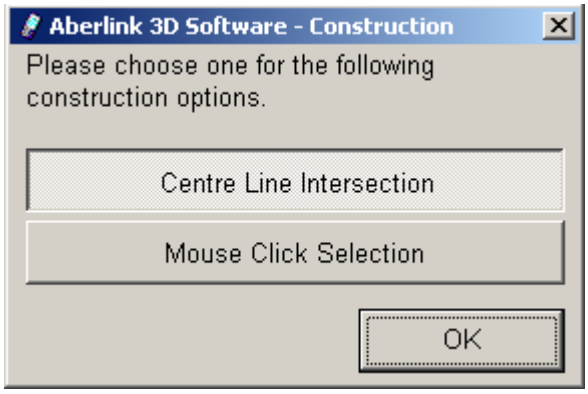


Fig. 12.2-8 Cylinder point construction 'Centre line options' window

www.EngineeringBooksPdf.com

After you have clicked on the 2nd cylinder the above options window will appear giving you two options select the first option 'Centre Line Intersection' This constructions a point were the centre line of the two cylinders meet.

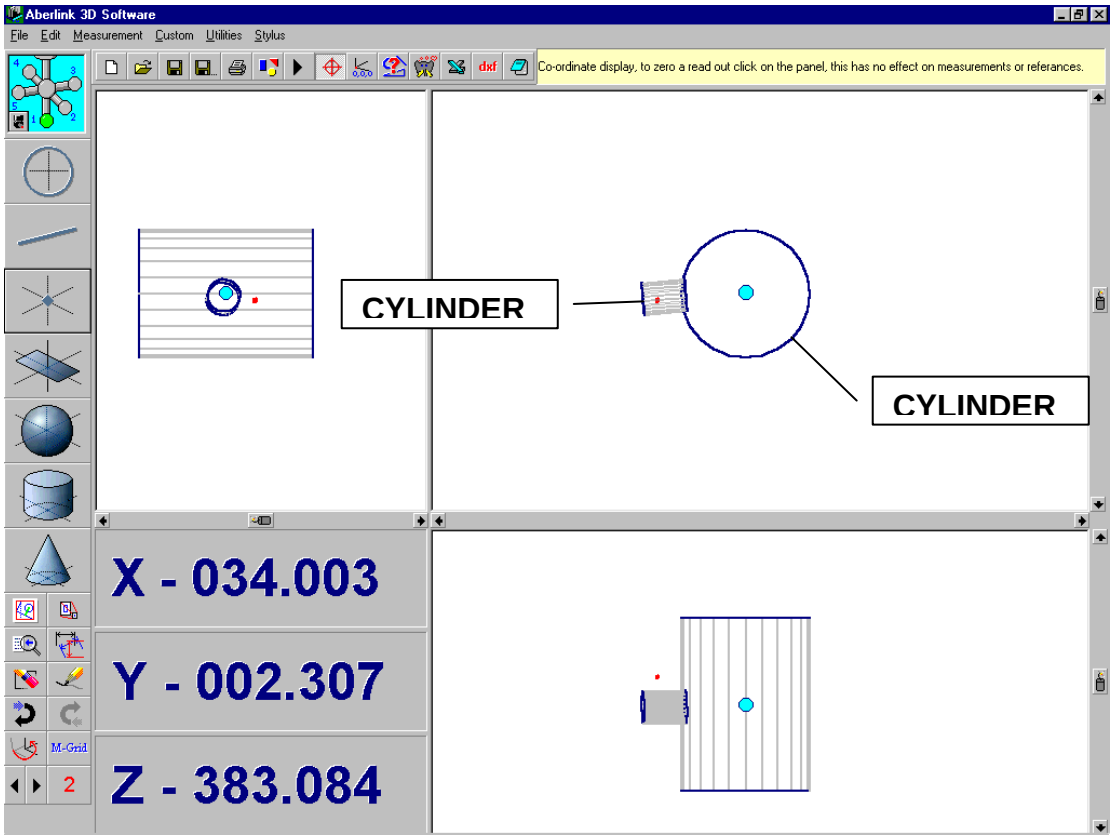


Fig. 12.2-10 Showing the centre line intersection point between 2 cylinders

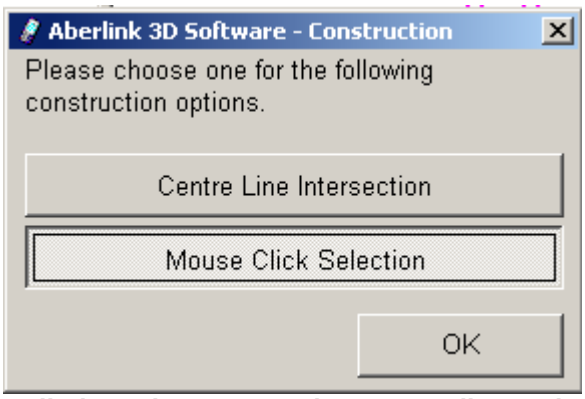


Fig. 12.2-8 Cylinder point construction 'Centre line options' window

Having constructed this first point, the order of selecting the cylinders for the remaining intersection points now has some relevance. The second intersection point will be positioned at the point where the second cylinder selected (cylinder B) intersects the wall of the first cylinder (cylinder A). After you have clicked on the second cylinder the options window will re appear this time take the 2nd option 'Mouse Click Selection' to construct a point where the second cylinder selected (cylinder B) intersects the left hand wall of the first cylinder (cylinder A) where you click on the 2nd cylinder will

www.EngineeringBooksPdf.com

determine if the point intersects the left hand wall or the right hand wall of the first cylinder.
So if we click on the left hand side of cylinder B you will get a point on the left hand side.
***Note** – Using the feature select function will allow you to always select a position close to the required intersection*

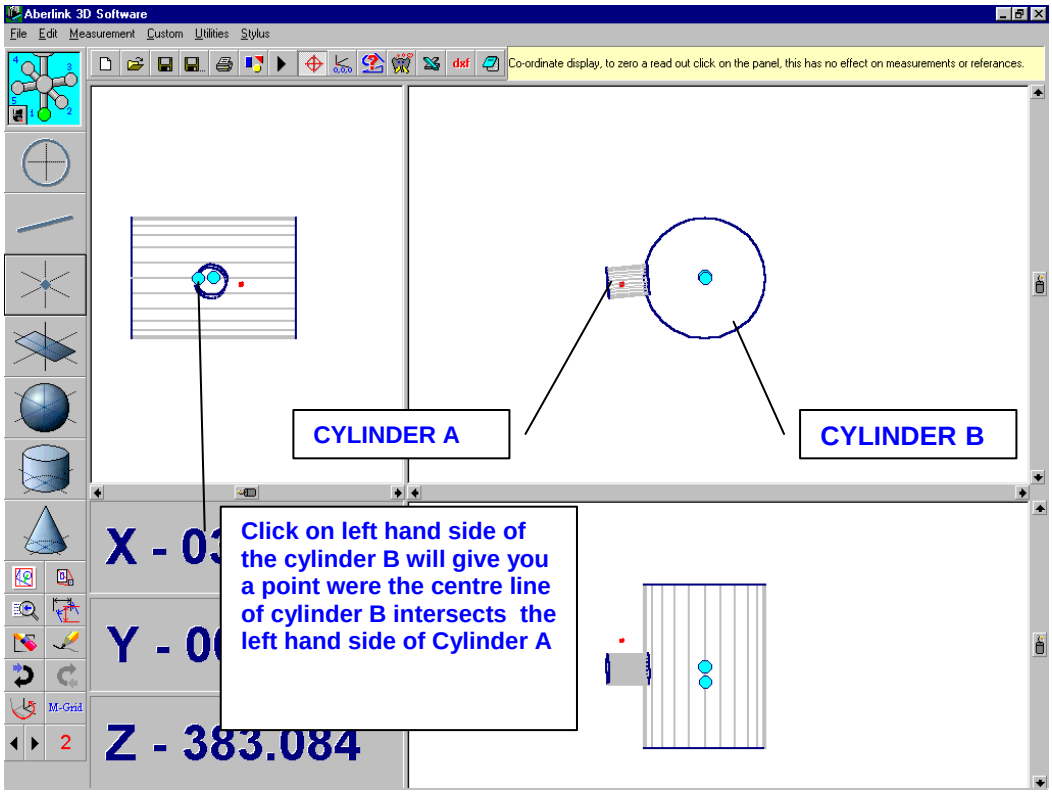


Fig. 12.2-11 Showing the second intersection point between 2 cylinders

If you now repeat this construction, selecting the cylinders in the same order but clicking on the right hand side of cylinder B , a second intersection point will be created where the centre line of the cylinder B intersects the right hand side of the cylinder A.

Now if you construct an intersection point, by selecting the cylinders in the reverse order The points you created will be where the centre line of cylinder A intersects with inner or outer walls of cylinder B.

www.EngineeringBooksPdf.com

To do this construction select cylinder B before A, this will construct fifth intersection point where the centre line of cylinder A intersect the wall of cylinder B

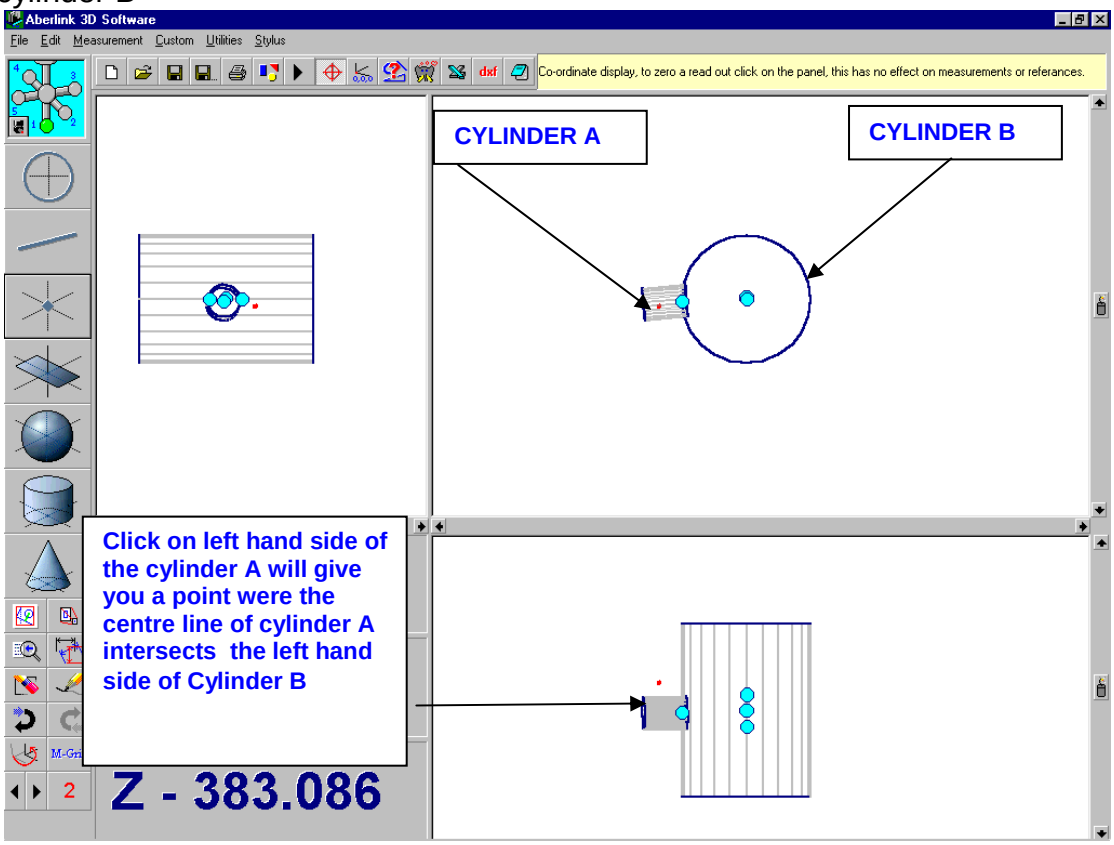


Fig. 12.2-13 Showing the fourth intersection point between 2 cylinders

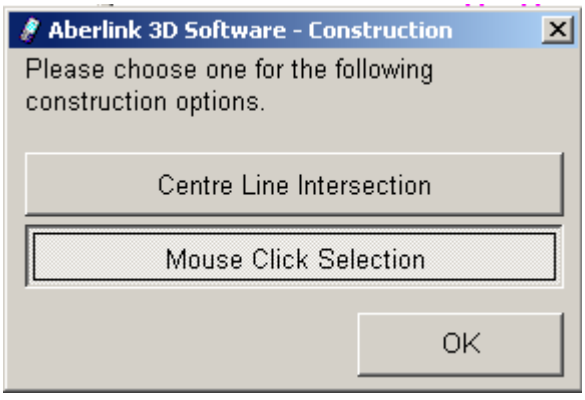


Fig. 12.2-8 Cylinder point construction 'Centre line options' window

As before after you have clicked on the second cylinder the options window will re appear this time take the 2nd option 'Mouse Click Selection' to construct a point where the second cylinder selected (cylinder A) intersects the wall of the first cylinder (cylinder B) where you click on the 2nd cylinder will determine if the point intersects the left wall or the right wall of the first cylinder

www.EngineeringBooksPdf.com

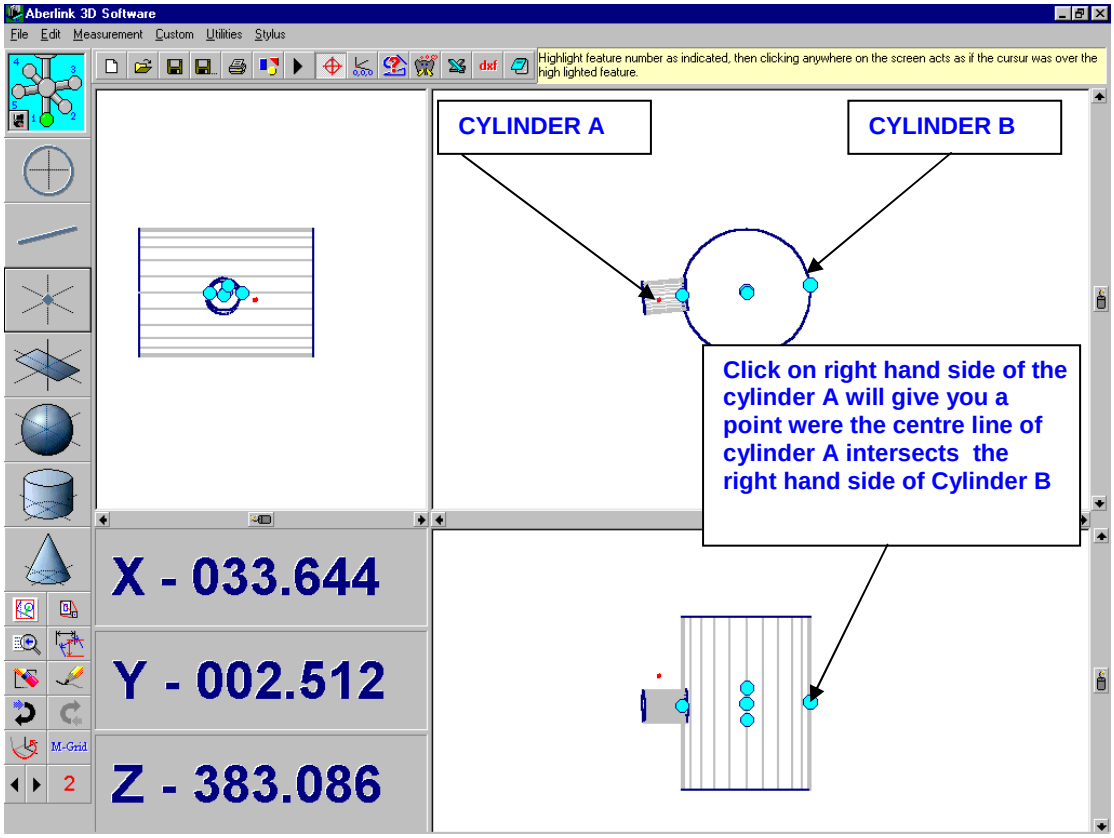


Fig. 12.2-14 Showing the fifth intersection point between 2 cylinders

12.2.5 Constructing a Point at the Mid-Point or the end of a Line

One further useful technique allows the construction of a point at the mid-point or the end of a line.

To construct such a point, click on the Point Measure button, from the main screen. This will bring up the Point Measure window. Now click on the 'Construct' box. A prompt will appear on the screen asking if you would like to enter the co-ordinates of the point. Click on 'No', and the Point Measure window will now shrink to a small box at the bottom of the screen.

Now simply select the line twice. After you have selected it for the first time, a prompt will appear, to confirm what is happening:

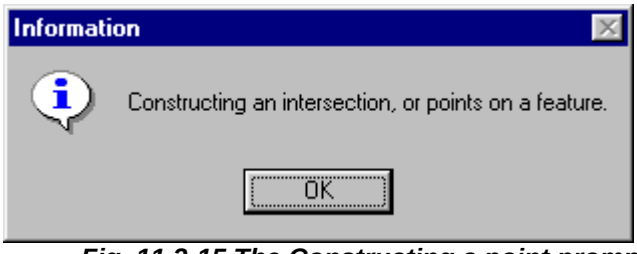


Fig. 11.2-15 The Constructing a point prompt

www.EngineeringBooksPdf.com

Click OK, and then click on the line a second time near it's centre. Now the Point Measure window will automatically reappear. The graphics part of the window and the Points Fit box will of course be blank, but the X,Y and Z co-ordinates of the constructed point will be displayed in red.

You may now click on 'OK' . The screen will now return to the Main Screen, and a point will have been constructed at the mid-point of the line.

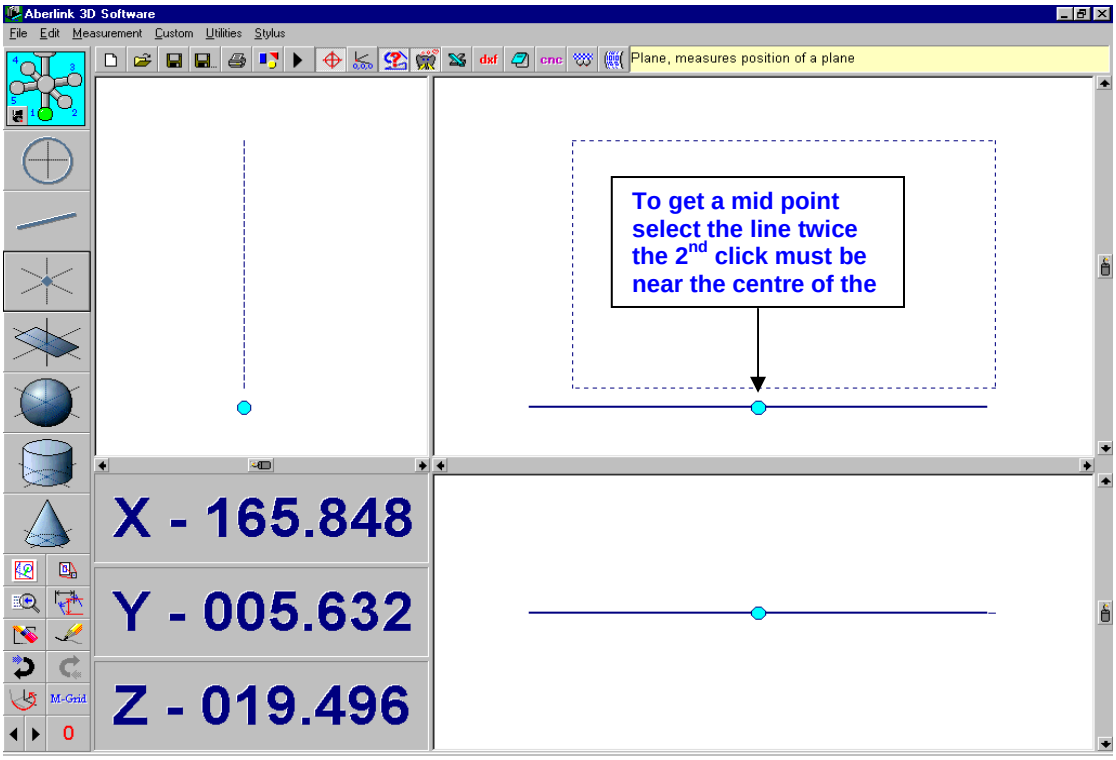


Fig. 12.2-16 A line with a point constructed at its mid-point

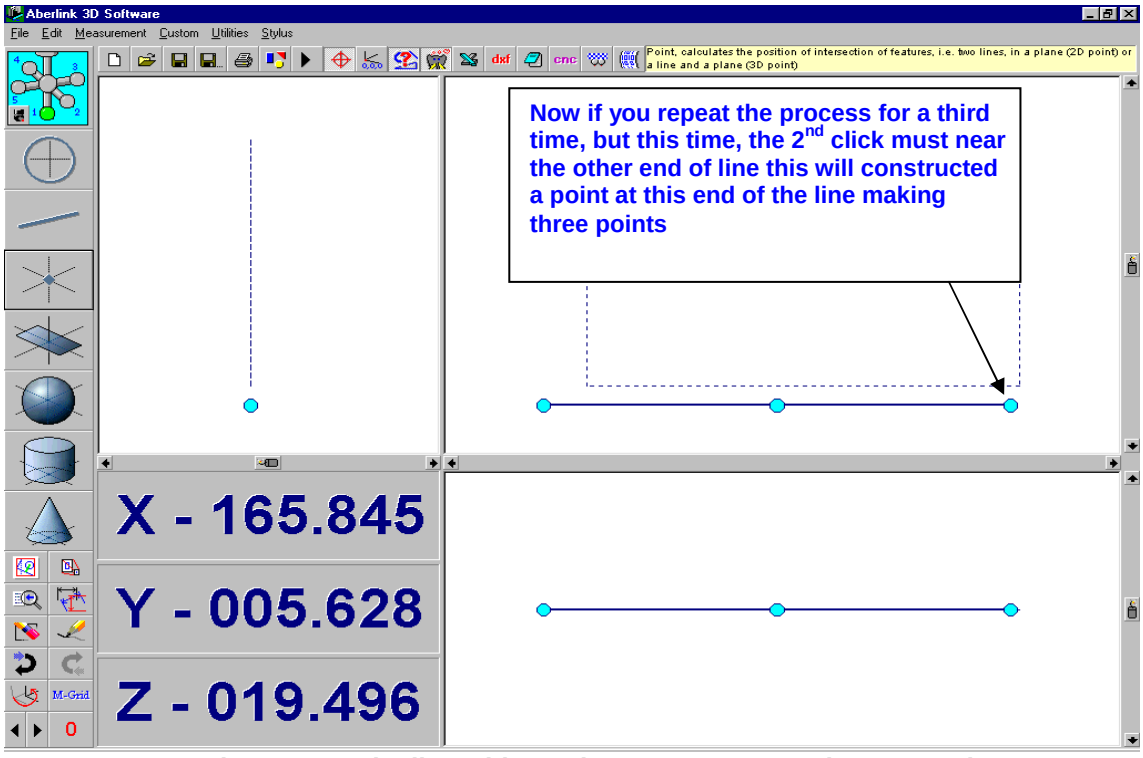
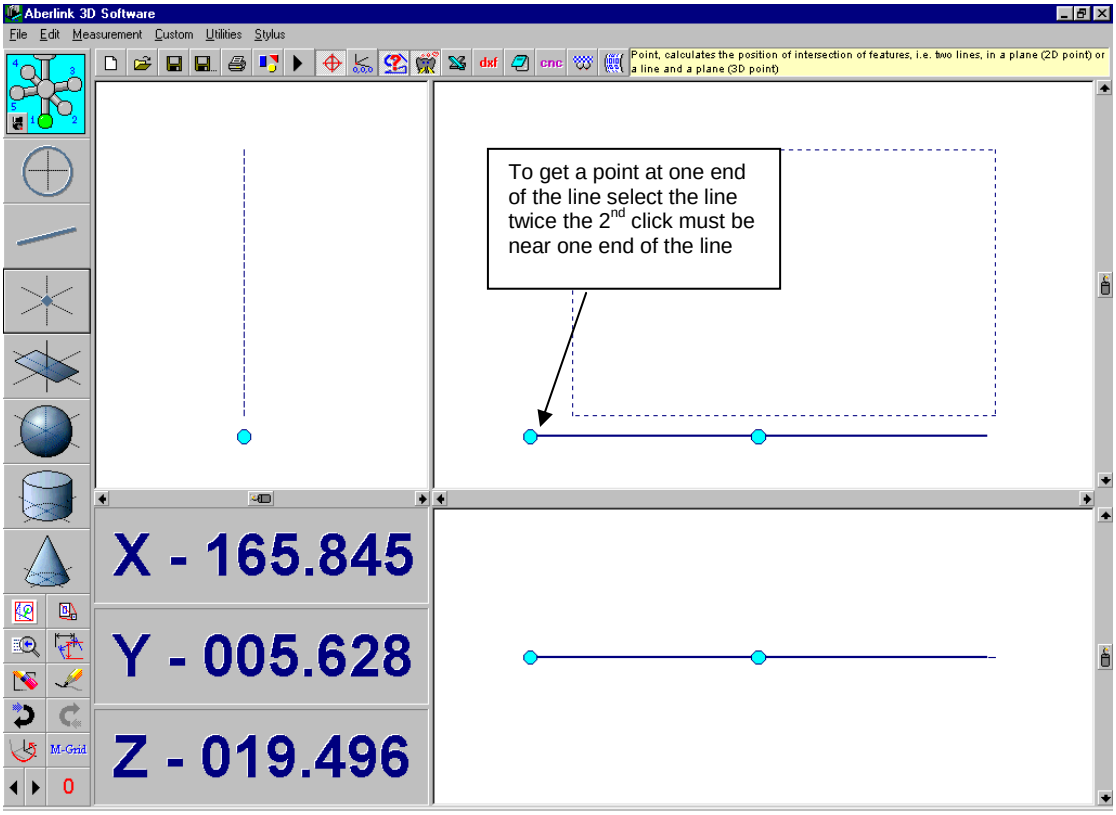


Fig. 12.2-17 The line with a point now constructed at one end

www.EngineeringBooksPdf.com

Now repeat exactly the same process but this time, the 2nd click must be near one end of the line this will constructed a point at one end.



\Fig. 9.2-18 The line with a point now constructed at the other end

Now if you repeat the process for a third time, but this time, the 2nd click must be near the end other end, this will constructed a point at the other end making three points.

www.EngineeringBooksPdf.com

Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

www.EngineeringBooksPdf.com

13.0 THE CYLINDER UNIT

The cylinder unit can be used to measure cylindrical shapes (holes or shafts etc.) by taking 5 or more measurement points on their surface.

Background

The software will construct the 'best fit' cylinder through the measurement points taken. In order to determine whether it is an internal cylinder (hole) or external cylinder (shaft etc.), and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.

The accuracy to which the software will calculate the best fit cylinder, and the maximum number of iterations that it will perform to achieve the best fit, can be set in the software set up – see chapter 26 – Software set up

As a cylinder is a 3 dimensional feature, no reference planes will need to be defined.

Method

To measure a cylinder, click on the Cylinder Measure button, from the main screen:

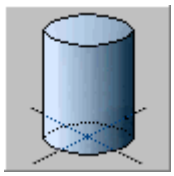


Fig. 13.0-1 The Cylinder Measure button

This will bring up the Cylinder Measure window:

www.EngineeringBooksPdf.com

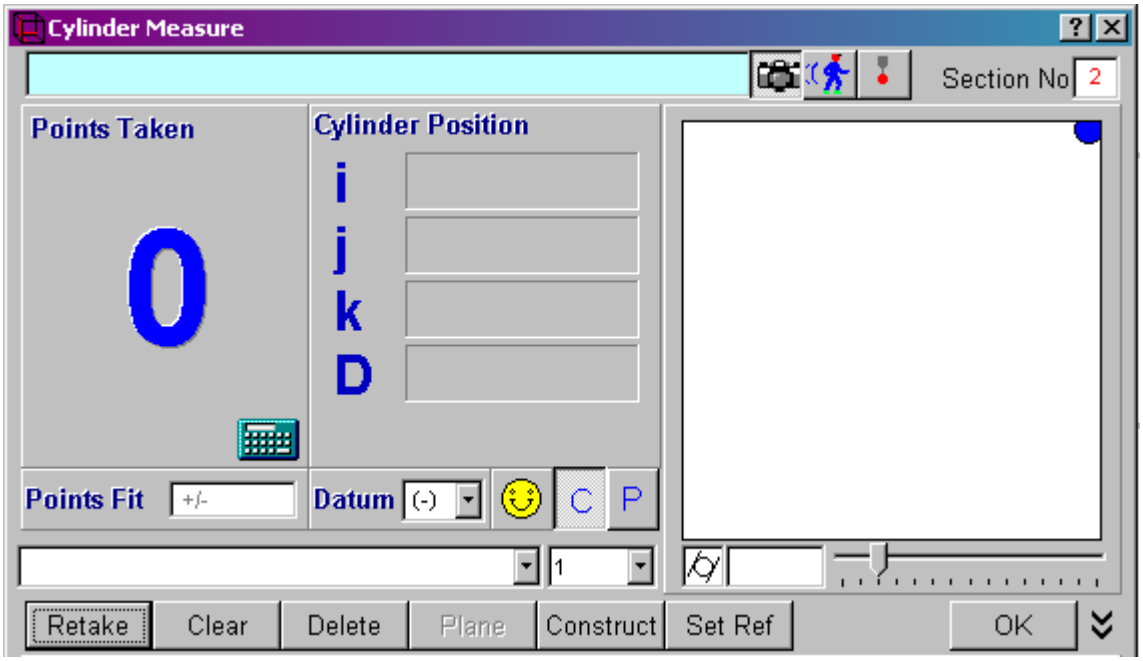


Fig. 13.0-2 The Cylinder Measure window

Now take measurement points on the cylinder that you wish to measure. After 5 points a pictorial representation of the cylinder will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The I,J and K values displayed represent direction vector for the cylinder, and the D value represents its diameter.

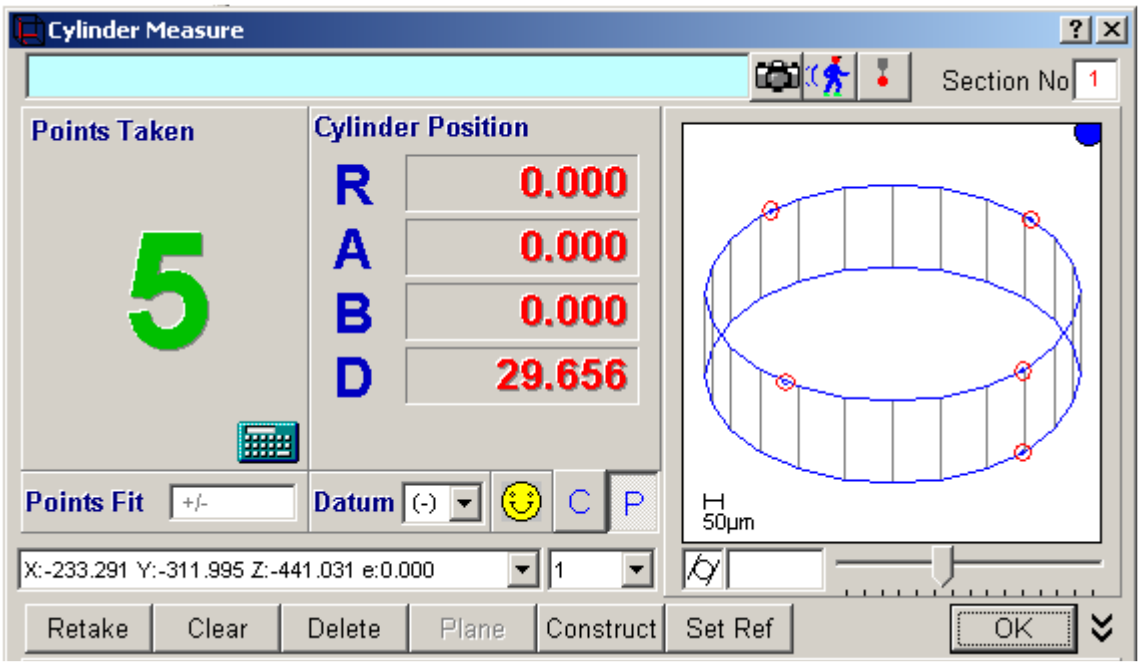


Fig. 13.0-3 A representation of a cylinder will appear after 5 points

The software requires a minimum of five points in order to calculate a cylinder, therefore the first 5 points will always produce single bleeps and the yellow face will remain smiling. Also the 'Points Fit' box next to the yellow face will

www.EngineeringBooksPdf.com

remain blank and the cylindricity value for the cylinder, shown under the graphical representation, will show zero.

It is therefore good practise to take at least one more point. On the sixth point the software will now be able to calculate a value for the fit of the points (2 sigma) on its calculated 'best fit' cylinder and a value for it's cylindricity. Also there is now some meaning to the 'bad point' sound and face warnings. If the last point taken doesn't fit the cylinder very well, then the face will start to frown. If it doesn't fit at all well, then the face will become sad, and the computer will make the 'bad point' sound. The threshold values for the smile and frown warnings can be user specified, see chapter 23 – Software set up.

In general the more points taken on a cylinder, the more information can be gleaned, and a minimum of 6 points is recommended. Similarly the points taken should cover as much of the surface of the cylinder as possible.

The graphical representation of the cylinder shows the points taken as a red circle with two blue circles joined by grey lines representing the best fit cylinder through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

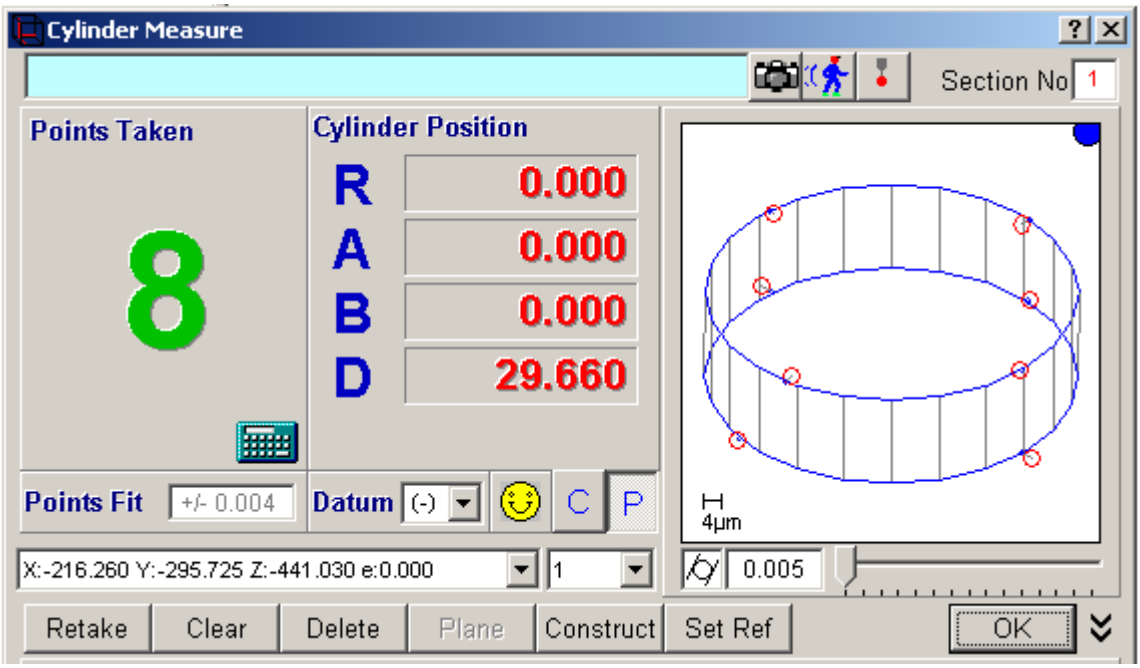


Fig. 13.0-4 A cylinder with the error scale maximised

www.EngineeringBooksPdf.com

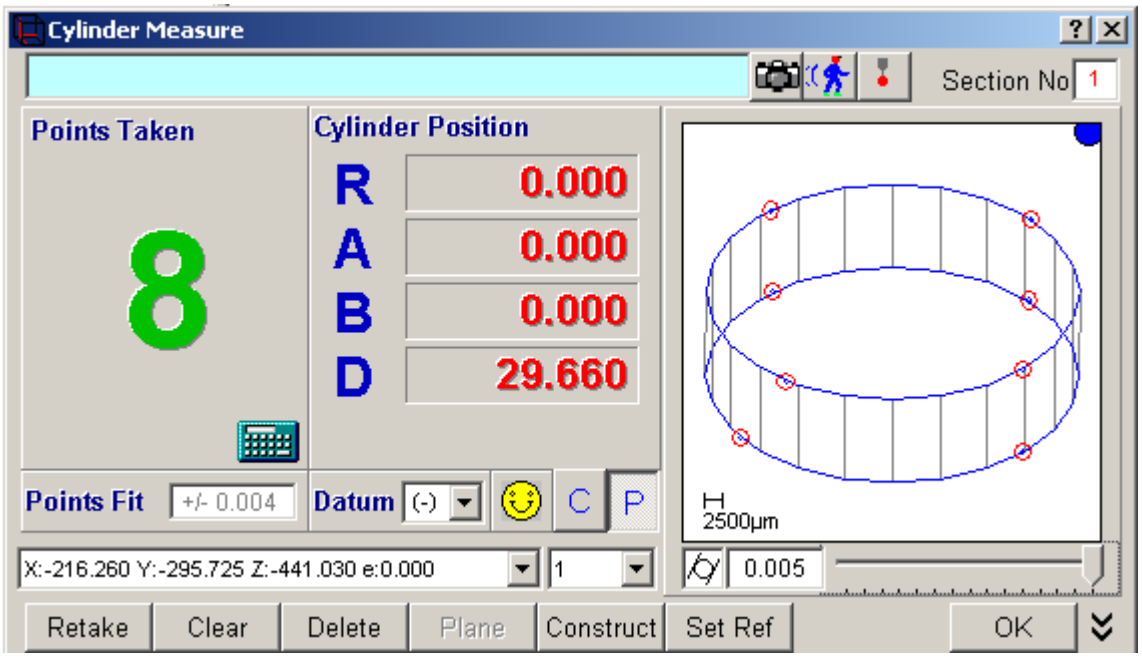


Fig. 13.0-5 A cylinder with the error scale minimised

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See chapter 14 - Erasing Data Points.

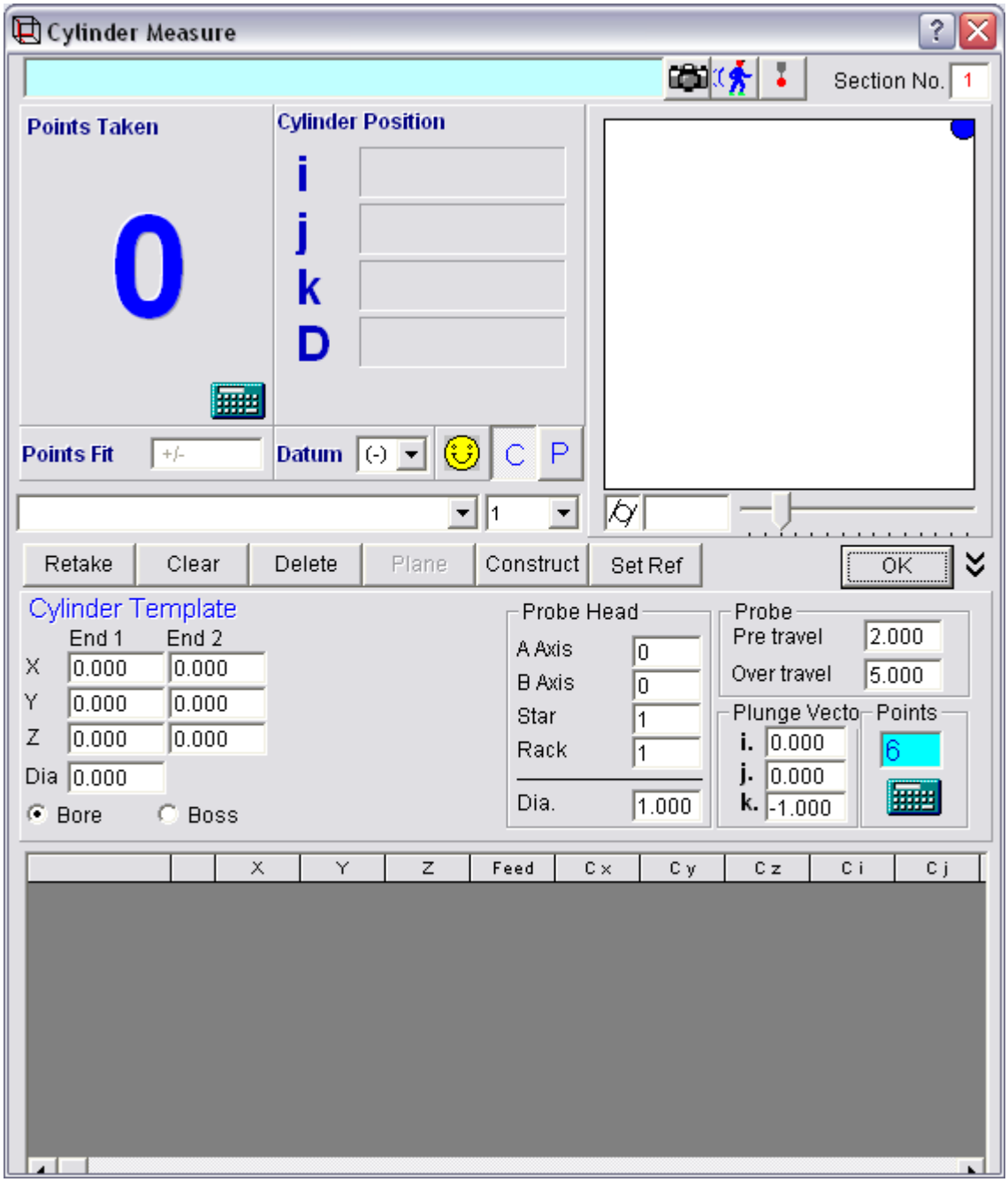
13.1.2 Off-Line Programming

Instead of directly measuring a cylinder using the probe it is possible to enter the co-ordinates for the cylinder and let the software calculate the measurement points to be taken uniformly around the cylinder.

In this situation note that the points calculated will always be evenly distributed around the cylinder, even if there are other features or obstacles in the way:

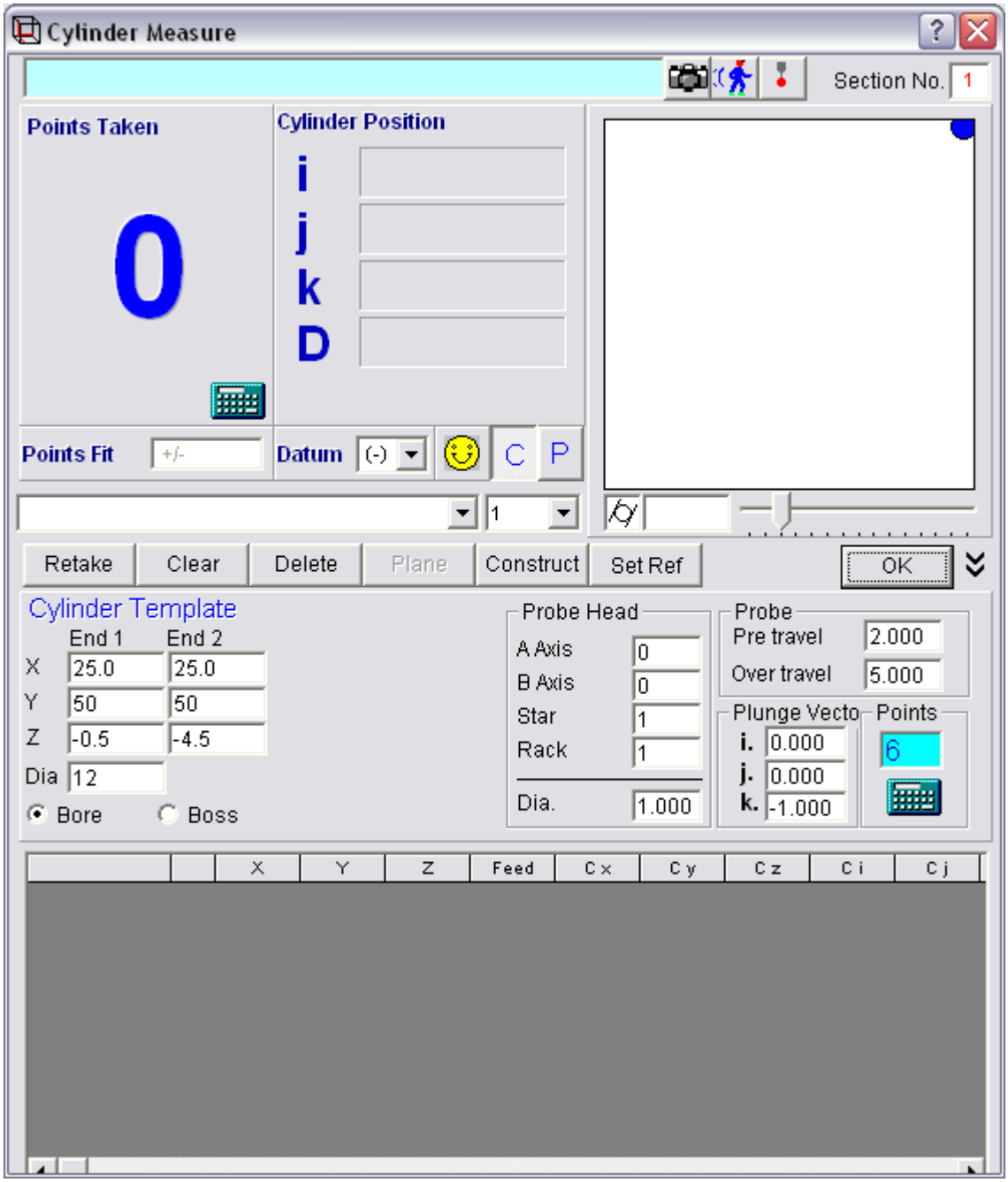
To create an off-line program for a cylinder click on the Cylinder Measure button, from the main screen to bring up the Cylinder Measure window. Now instead of taking points with the probe we are going to type in the template details directly:

www.EngineeringBooksPdf.com



A cylinder is defined by entering the X,Y,Z co-ordinates of both ends of its centre line and its diameter. Assume we have a flat bottomed hole that we wish to measure as a cylinder at X=25, Y=50, Z=0 and its diameter is 12mm. The depth of the hole is 6mm and we want to use a 2mm stylus. Enter the co-ordinates for the cylinder into the Cylinder Template section of the window, as follows:

www.EngineeringBooksPdf.com

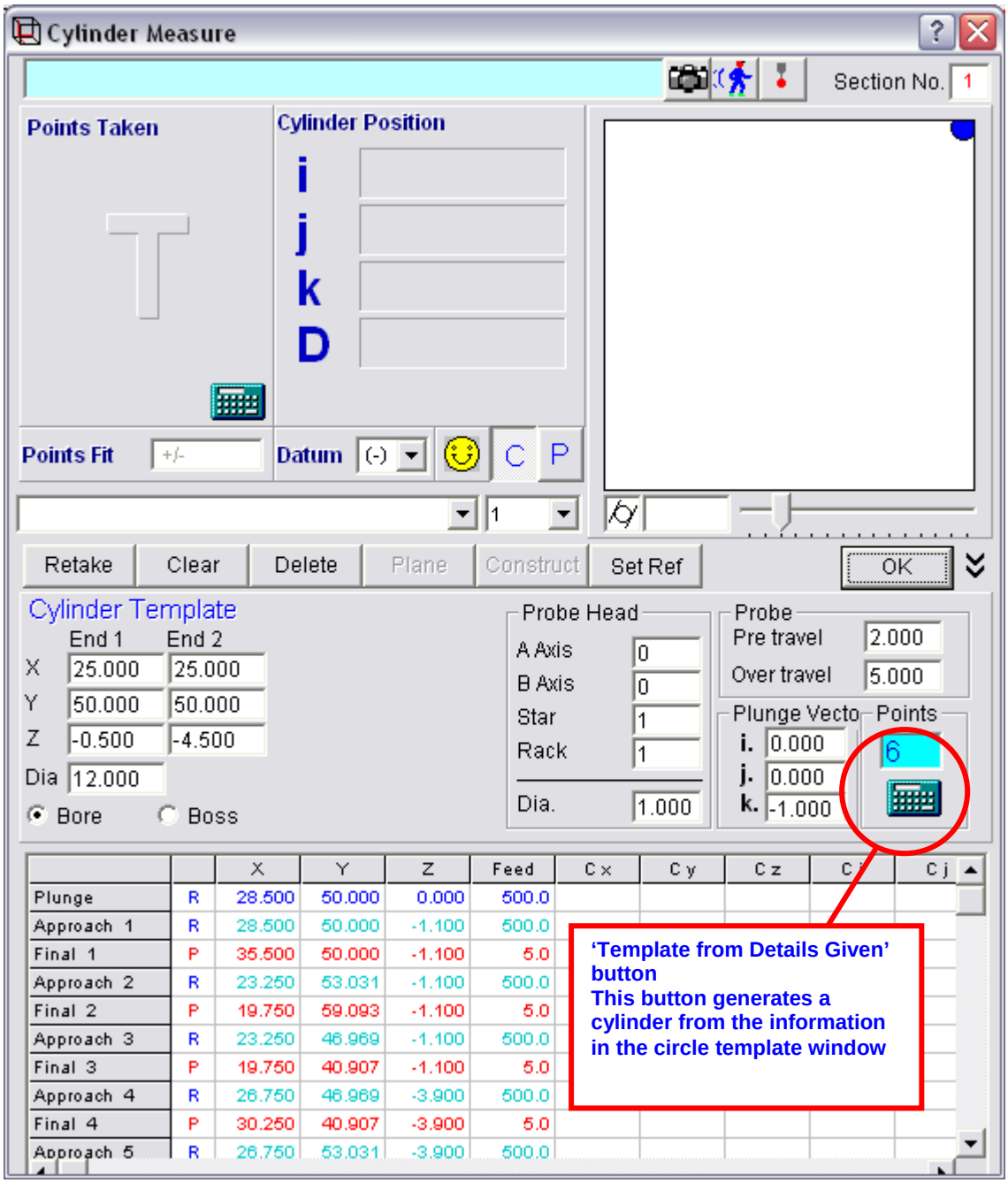


Make the top end of the cylinder at Z=-0.5 so that all points are taken with the centre of the stylus 0.5mm below the surface (you may need to increase this value if the hole has a larger chamfer, for instance). Make the bottom of the cylinder at Z=-4.5 . This allows 1mm for the stylus radius plus 0.5mm clearance at the bottom of the hole. Enter the diameter at 12.

The pre-travel and the over travel will be set at their default values, but can be modified as required. Similarly you can change the probe head position and plunge vector (direction from which the probe will approach the cylinder) if required. Note that the default vector is defined by the orientation of the probe at the time of programming. The default number of points on the cylinder has been set to 6 – lets leave it at 6 for the moment.

www.EngineeringBooksPdf.com

Now click on the ‘Template from Details Given’ button. The probe moves grid will now be automatically filled in, and the 6 points will be positioned around the cylinder defined:



You can now click on OK (or Set Ref and OK) as desired. The cylinder will appear as a grey template in the main screen.

Say that we now changed our mind and decided that measurement of the cylinder was far more critical than at first thought, and we now wanted to take 10 points. Recall the Cylinder Measurement window (by right clicking on the representation of the cylinder) and change the number of points to 10. Now click on the ‘Template from Details Given’ button again and the probe moves

www.EngineeringBooksPdf.com

grid will be re-calculated now with 10 points in it. Click on OK to return to the main screen.

If you click on the 'Display Probe Contact Points' button the measurement points will be displayed as dots:

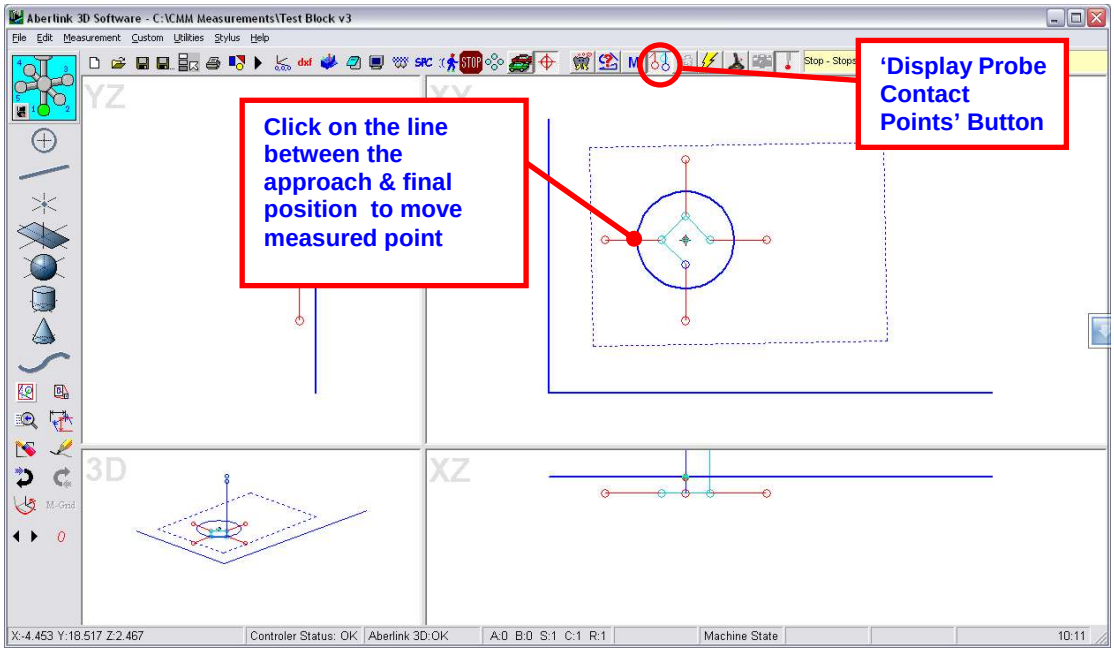


Fig. 8.1.2-4

If any of the points on the cylinder were in an inappropriate position (for instance they coincided with a slot or other feature) there are 2 courses of action available:

3. Move the point to a more suitable position

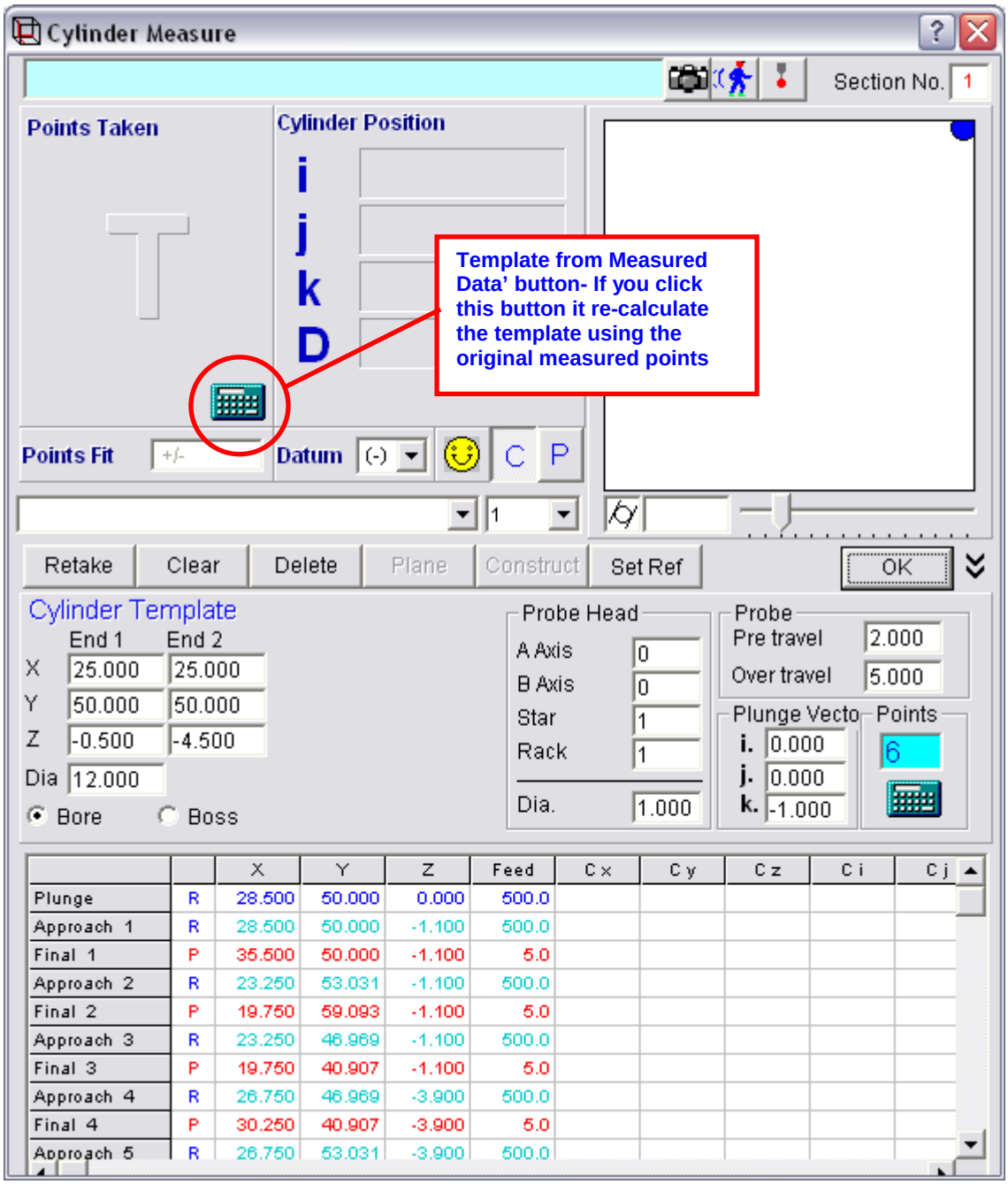
With the 'Display Probe Contact Points' button pressed so that the measurement points are visible, simply click on the measurement line that connects the approach point to the final point and drag it to a convenient position on the circumference of the cylinder. When you release the mouse button the point will snap to the closest position on the cylinder from where you release it. See Chapter 40 – The 'Display Probe Contact Points' function

4. Erase the point

In this case recall the Circle Measure window and from the probe moves grid delete the 2 lines that go to make up the point. For instance if you wish to erase point number 3, click on the box that says 'Approach 3', then right click and select 'Delete Cells'. Repeat the process for the move called 'Final 3'.

www.EngineeringBooksPdf.com

13.1 3 The 'Template from Measured Data' button



When you create a program unit using the 'teach and repeat' method the program template is automatically calculated from measurement points taken.

If however, you need to re-calculate the template at any time, but cannot use the template details (for instance the points cannot be equi-spaced) then it is possible as follows:

- Clear the existing points, by clicking on the 'Clear' button.
- Take some new measurement points with the probe.
- Click on the 'Template from Measured Data' button to create a new program template from the points taken.

www.EngineeringBooksPdf.com

Note when you click on the ‘Template from Measured Data’ button the probe moves grid is re-calculated using the points taken. These points are shown in the top left hand corner of the window.

It is bad practice to re-calculate the template for a feature that has been set as a reference once the program has been run.

13.1.4 When to OK a Cylinder, and when to reference it.

Having measured a cylinder, you can either click on the ‘OK’ button or the ‘Set Ref ‘ button to reference the cylinder.

If you OK the cylinder, the measurement window will close, and the program will return to the main screen.

If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position. See chapter 14 - Manipulating the Graphics.

The graphical representation of the cylinder should be shown as two solid blue lines representing the ends of the cylinder, with grey lines joining them. This should be visible in all of the 3 views

www.EngineeringBooksPdf.com

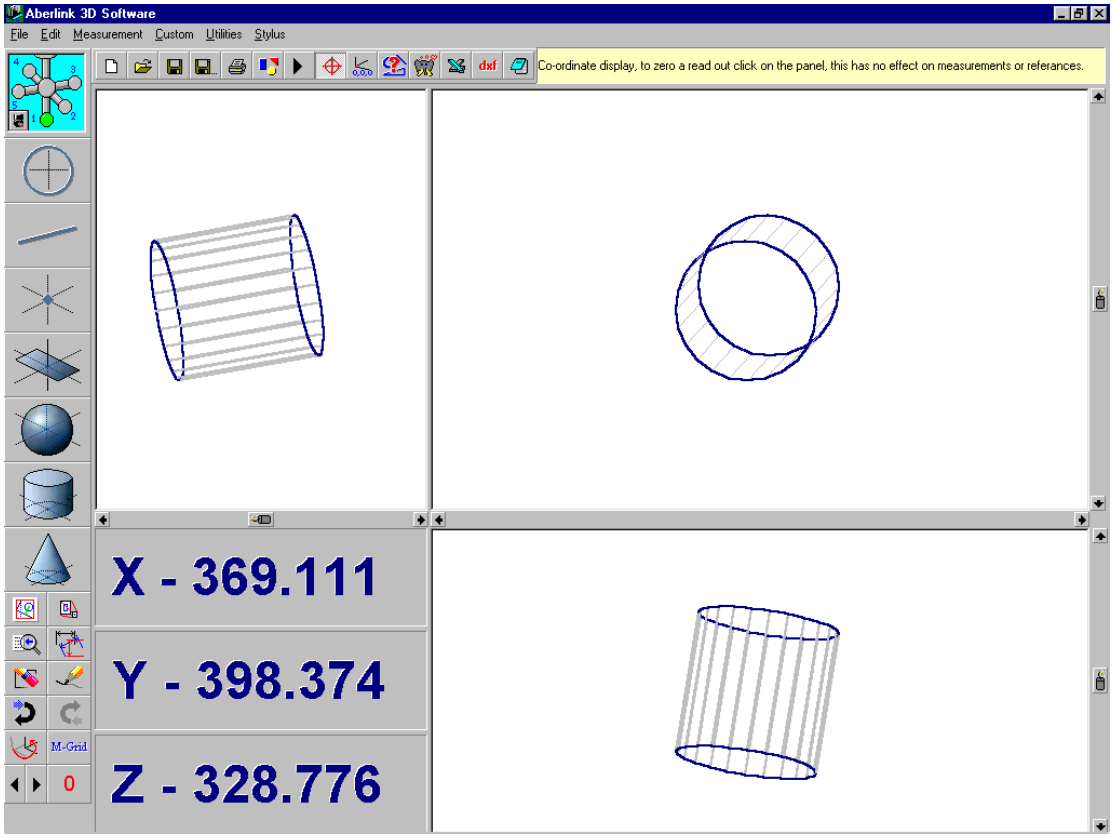


Fig. 13.1-1 The Main Screen showing a cylinder that has been OK'd

The angle at which the cylinder is shown will be relative to any previous references that have been set, or relative to the machine axes if no references have been set.

You can set the cylinder as a reference by clicking on 'Set Ref' button.

If a plane has previously been measured and set as a reference, the cylinder will be used to define 2 further degrees of freedom (see chapter 6 – Aligning the Axes of a Component), and will define one of the orthogonal planes to the reference plane.

www.EngineeringBooksPdf.com

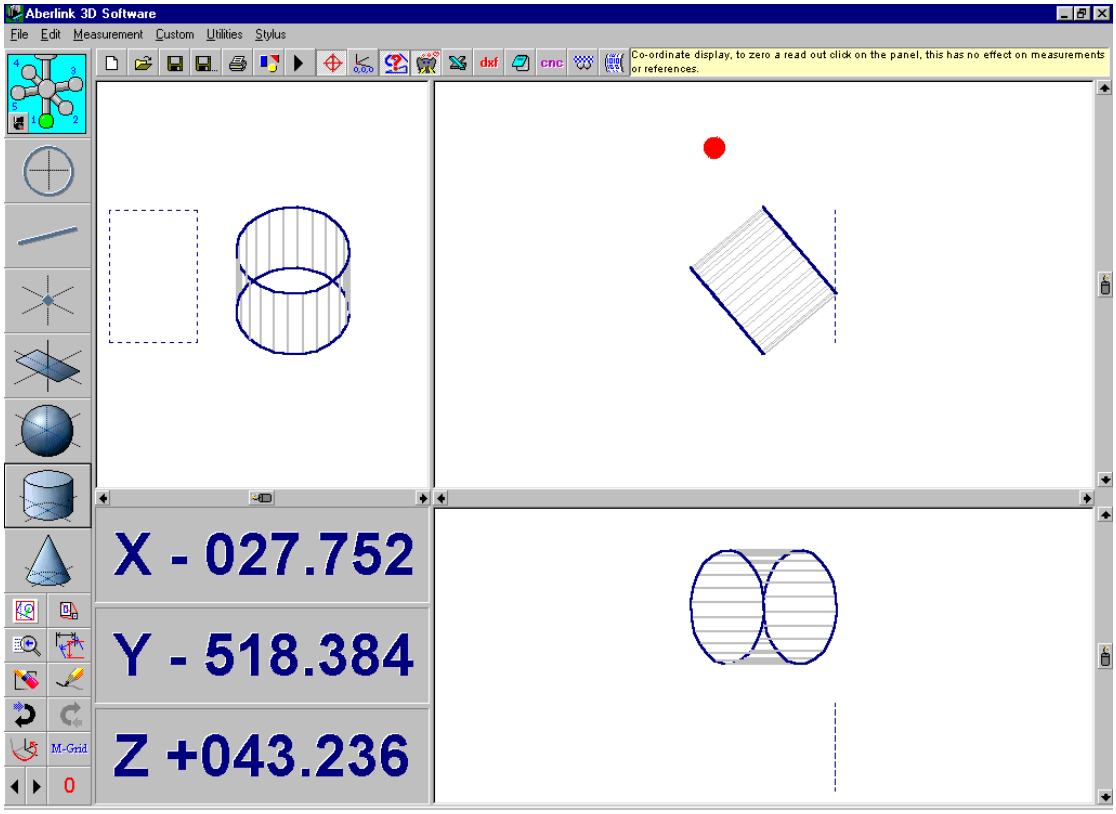


Fig. 13.1-2 A referenced cylinder with a reference plane

If however, no reference plane has previously been set, the cylinder will be aligned to one of the axes, thus defining 4 degrees of freedom.

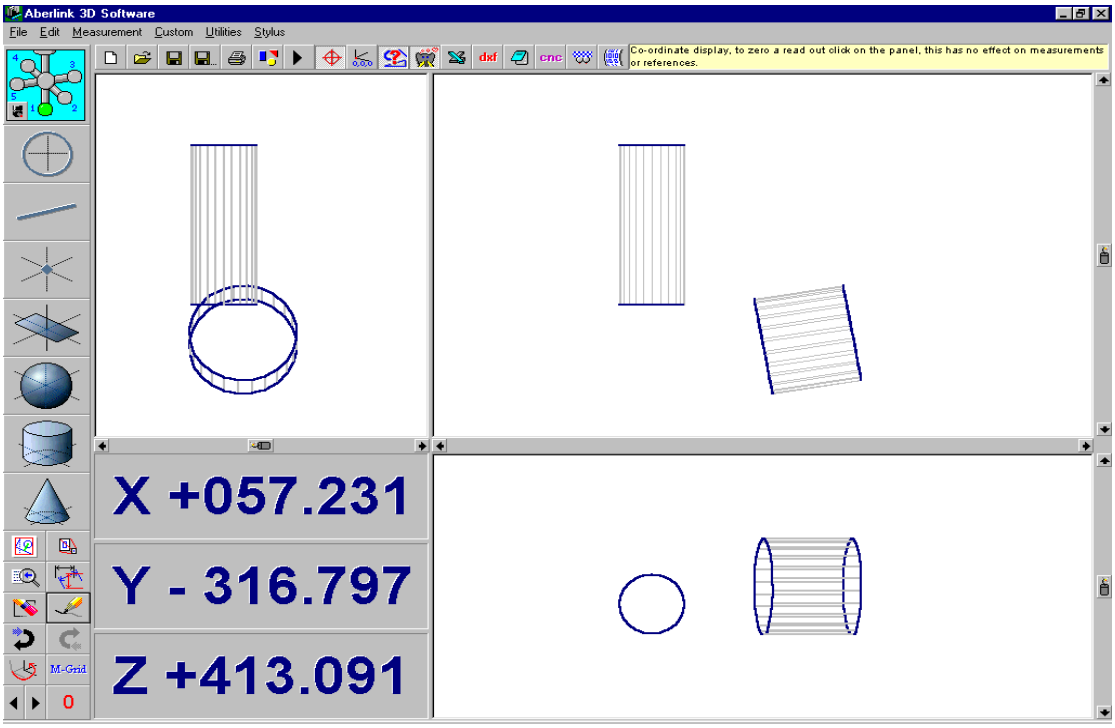


Fig. 13.1-3 A referenced cylinder with no reference plane

www.EngineeringBooksPdf.com

If a second cylinder is then set as a reference, this cylinder will supersede the first as the primary reference, but the original cylinder will still be used to define any degrees of freedom that are not defined by the second cylinder.

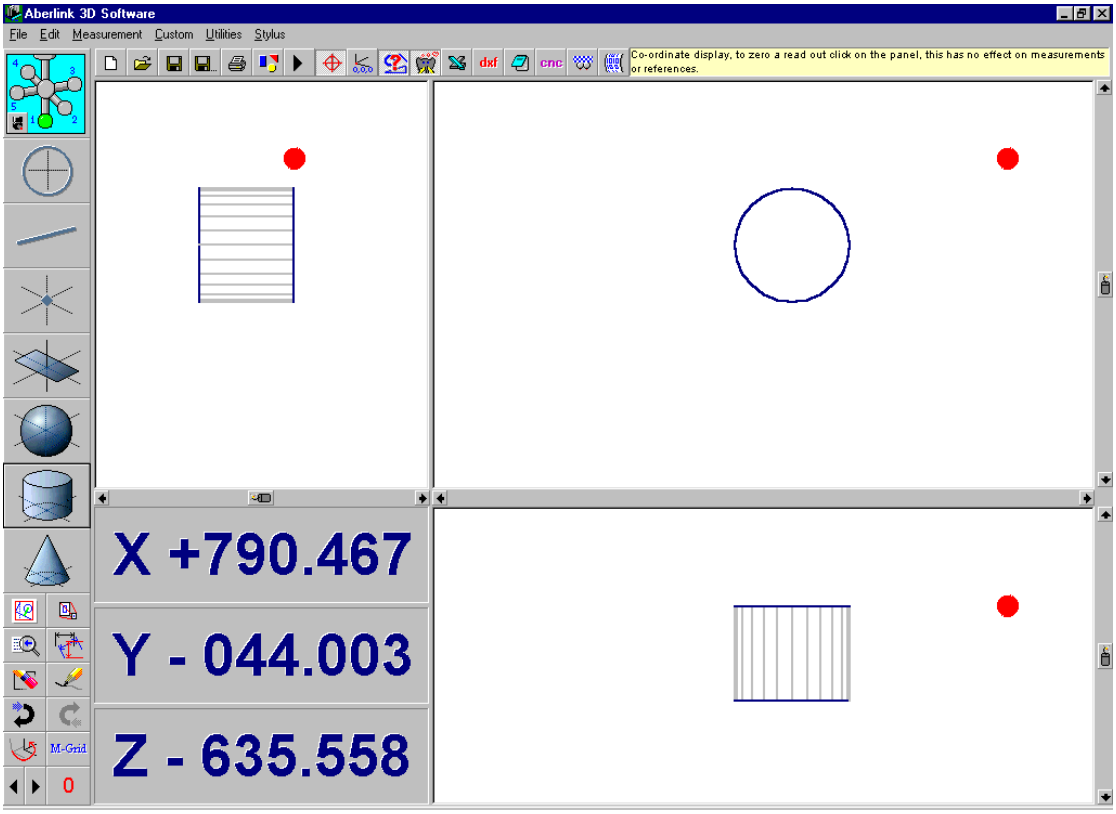


Fig. 13.1-4 Two cylinders that have been referenced

In this case the second, longer cylinder is now aligned with the Y axis, and the original cylinder is now orientated to define the XY plane.

www.EngineeringBooksPdf.com

14.0 THE CONE UNIT

The cone unit can be used to measure conical features such as countersinks and circular chamfers. The method required to measure a cone, and the number of measurement points needed, depends upon whether the user is able to define the axis for the cone before taking the measurement points.

Background
The software will construct the 'best fit' cone through the measurement points taken.

If the user is not able to define the axis of the cone before starting the measurement, then the measurement points must be taken in a specific order. A first set of measurement points must be taken near one end of the cone, followed by a second set at the opposite end of the cone. The number of measurement points in a set can be defined by the user, (the default number is defined in the software set up – see chapter 23 – Software set up), but must be a minimum of 3.

If the user is able to define the axis of the cone, then the measurement points may be taken in any order, and a minimum of 6 points is required.

In order to determine whether it is an internal or external cone, and apply the correct stylus radius offset, the software will look at the direction of motion of the probe when taking the measurement points.

14.1 Measuring cones without defining their axis

Method
Click on the Cone Measure button, from the main screen:

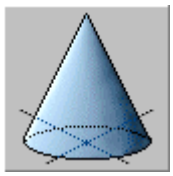


Fig 14.1-1 The Cone Measure button

This will bring up the Cone Measure window:

www.EngineeringBooksPdf.com

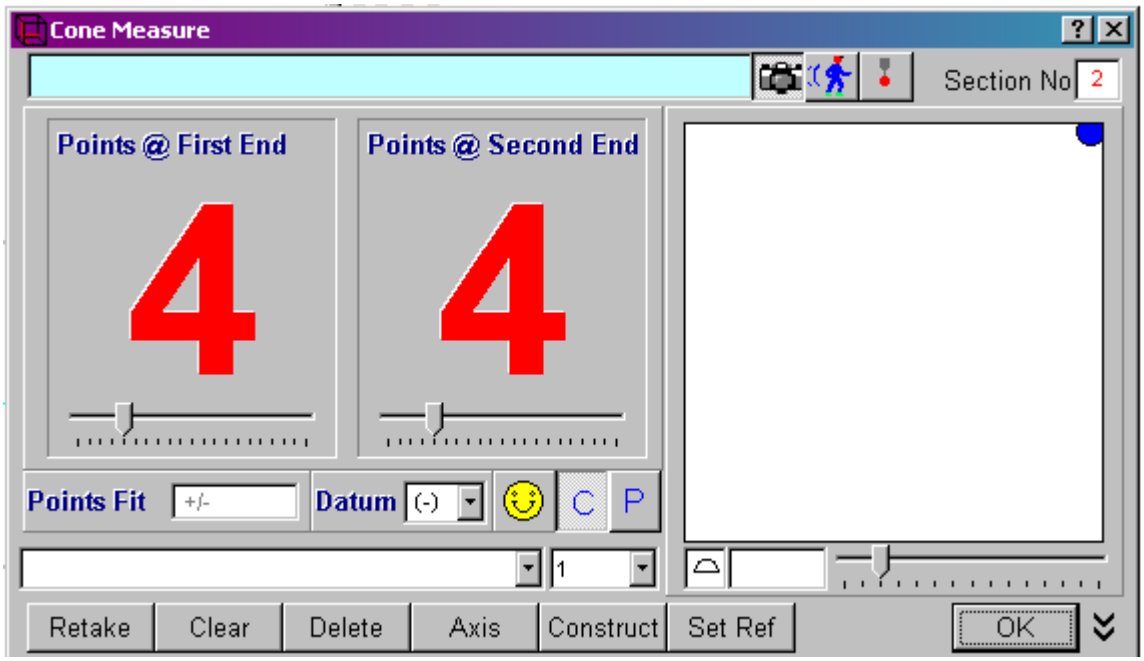


Fig 14.1-1 The Cone Measure button

Note that instead of having one large figure on the screen displaying points taken, there are now two figures on the screen, one under the heading ‘Points at First End’ and one under the heading ‘Points at Second End’. These numbers can be adjusted using the slider bar in each box, and the default number can be set in the software set up – see chapter 23 - Software set up.

The split box denotes that it is important to take some measurement points (minimum 3) near one end of the cone, and some points (again minimum 3) at the other end. It is important to try to keep the depth of these sets of measurement points consistent, and also to try to spread the two different sets of points as far apart as possible.

Each time you take a measurement point at the first depth in the cone, the number under ‘Points at First End’ will reduce by 1, until it is zero. You must then start to take points at the other depth on the cone. Now for each point taken the number under ‘Points at Second End’ will reduce by 1.

When you have reduced both numbers to zero, and providing that the software can calculate a cone to the required accuracy, then the window will revert to the usual single figure of ‘Points Taken’ and a pictorial representation of the cone will appear on the graphics part of the window.
The X,Y and Z values displayed represent the co-ordinate for the apex of the cone, and the A value represents its included angle.

www.EngineeringBooksPdf.com

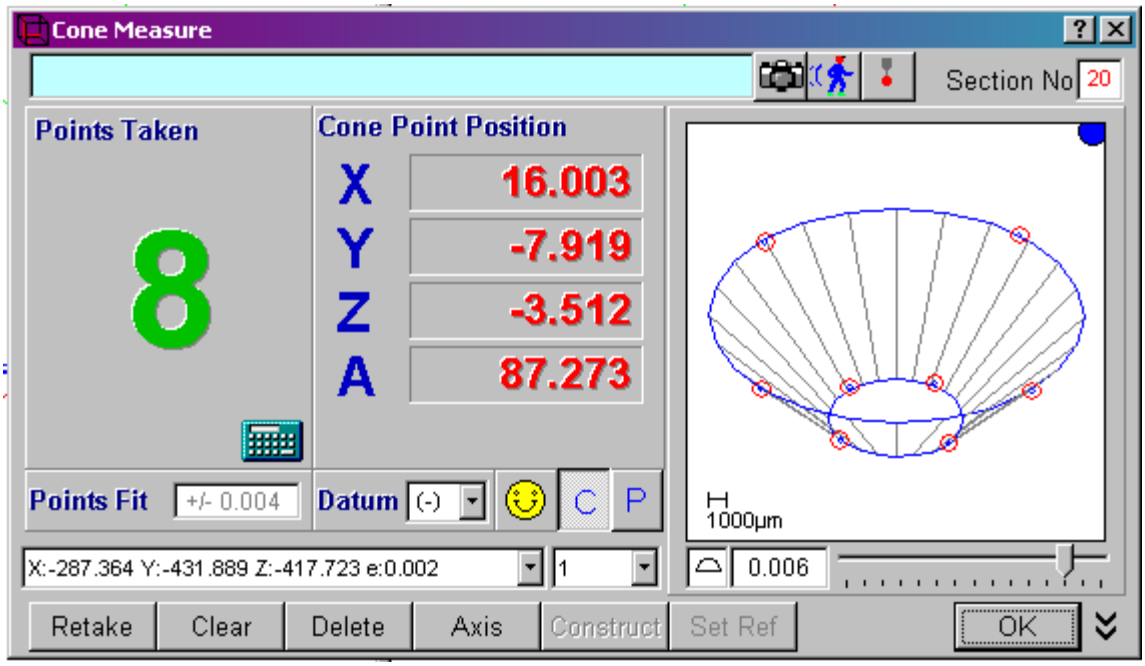


Fig. 14.1-3 The cone will appear once both sets of points have been taken

You may now continue to take extra measurement points at any depth on the cone. After each point the software will update the graphical representation and calculate fresh values for the fit of the points on the calculated 'best fit' cone, and for the profile of the cone. Also there is now some meaning to the double bleep and face warnings, and if any of the points that have been taken were 'bad' points this will now be highlighted.

In general the more points taken on a cone, the more information can be gleaned, and a minimum of 6 points is required. Similarly the points taken should cover as much of the surface of the cone as possible.

The graphical representation of the cone shows the points taken as a red circle with a blue circle and a point joined by grey lines representing the best fit cone through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

www.EngineeringBooksPdf.com

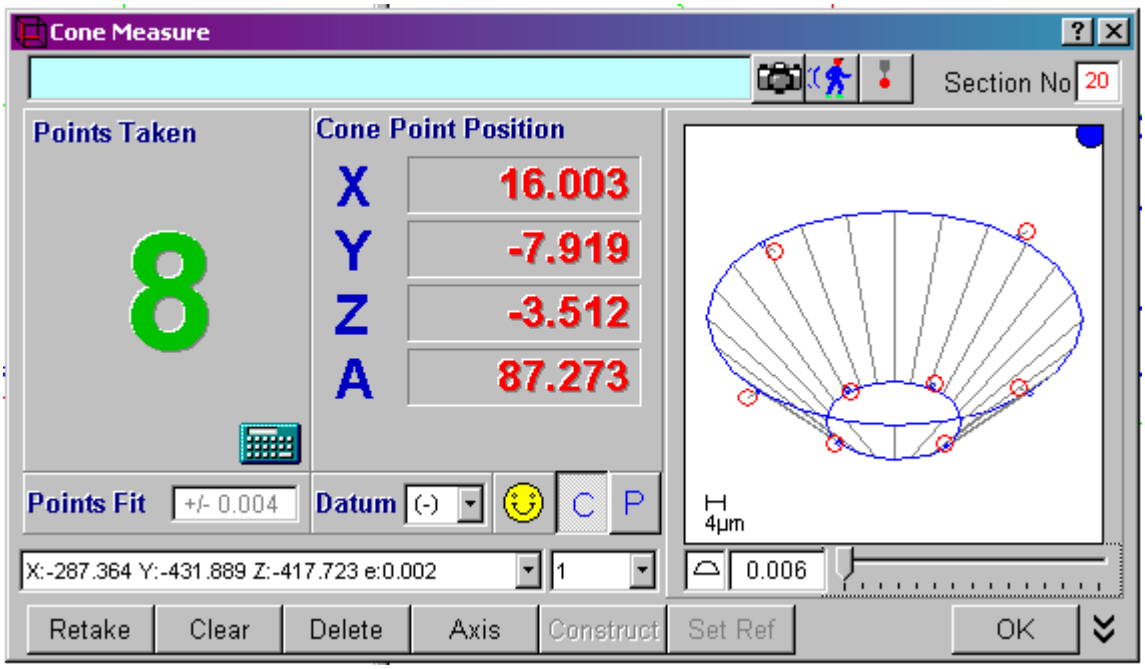


Fig. 14.1-4 A cone with the error scale maximised

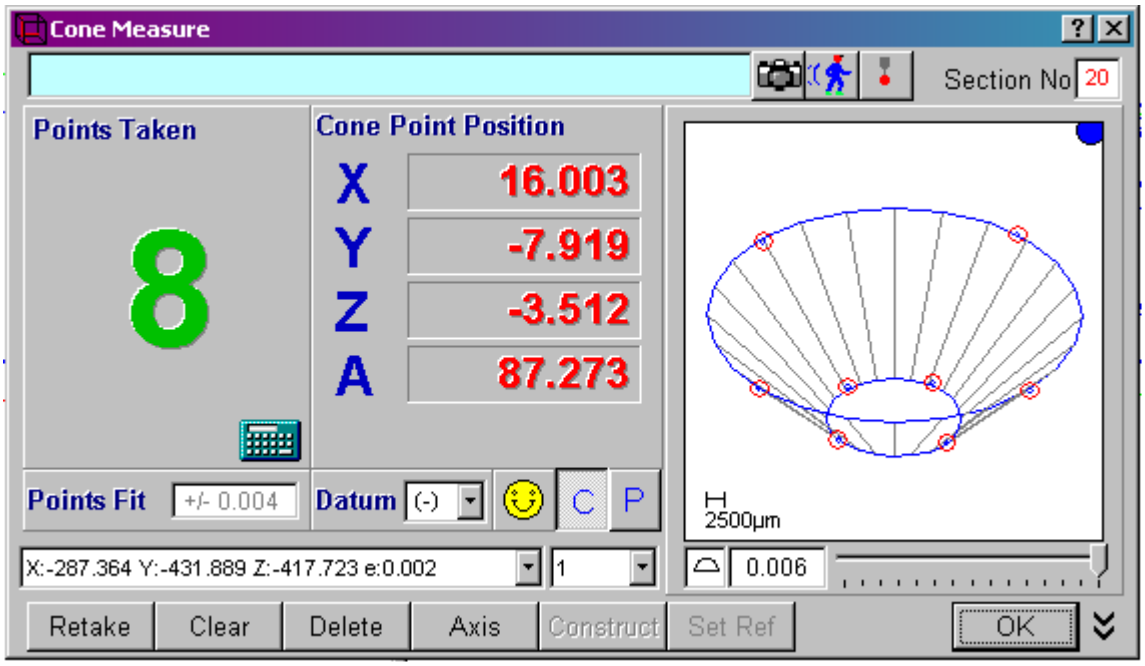


Fig. 14.1-5 A cone with the error scale minimised

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See chapter 13 - Erasing Data Points.

www.EngineeringBooksPdf.com

14.2 Measuring cones having defined their axis

Method

Click on the Cone Measure button, from the main screen. This will bring up the Cone Measure window. Now click on the 'Axis' button, and the following window will appear:

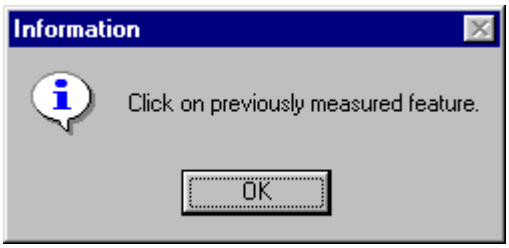


Fig. 14.2-2 The prompt when defining a User Defined plane

You can now define the direction of the axis of the cone, Click the 'OK' button in the above prompt window.
And then click on a previously measured feature to define the axis of the cone.

The measurement window will revert to the usual Zero figure for 'Points Taken', and you can now proceed to take measurement points on the cone in any order.

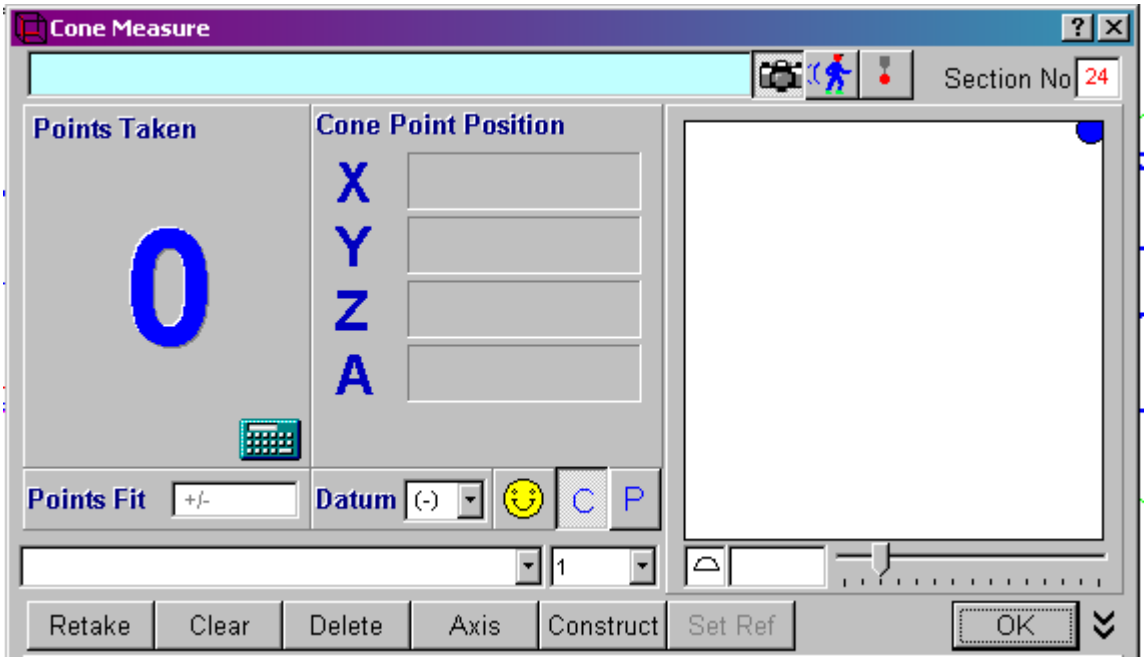
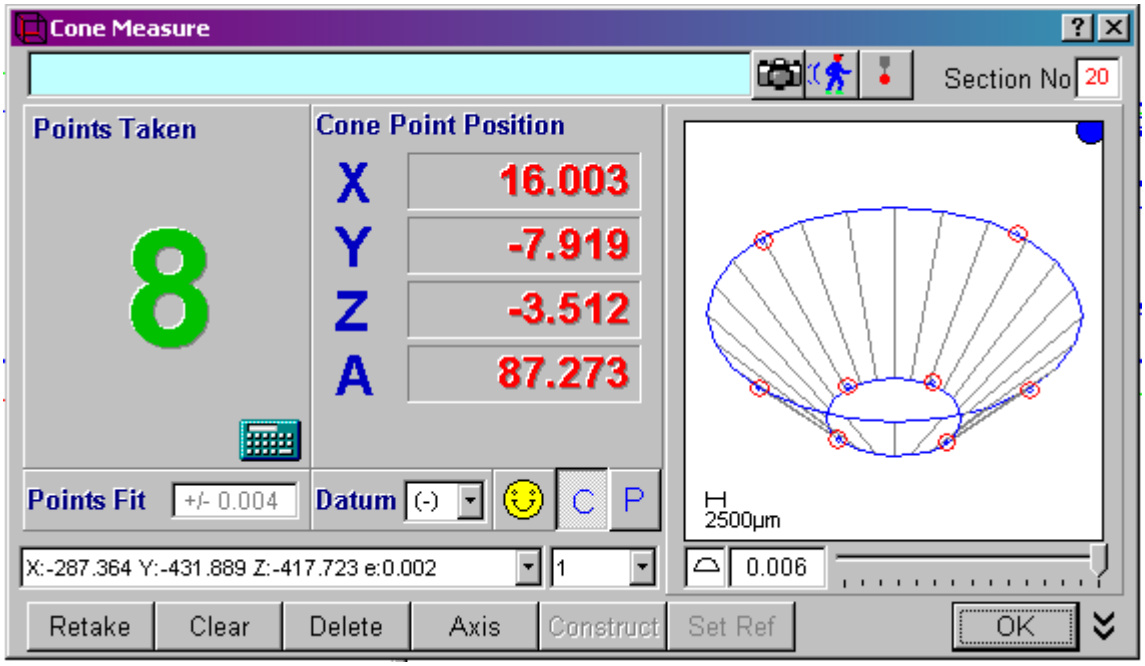


Fig. 14.2-3 The Cone Measure window having defined the axis of the cone

After 6 points, providing that the software can calculate a cone to the required accuracy, a pictorial representation of the cone will appear in the graphics part of the window, which will be updated every point thereafter. The number of points taken will be shown as the large character in the 'Points Taken' box. The X,Y and Z values displayed represent the co-ordinate for the apex of the cone, and the A value represents its included angle.

www.EngineeringBooksPdf.com



You may now continue to take extra measurement points at any depth on the cone. After each point the software will update the graphical representation and calculate fresh values for the fit of the points on the calculated ‘best fit’ cone, and for the profile of the cone. Also there is now some meaning to the double bleep and face warnings, and if any of the points that have been taken were ‘bad’ points this will now be highlighted.

In general the more points taken on a cone, the more information can be gleaned, and a minimum of 6 points is required. Similarly the points taken should cover as much of the surface of the cone as possible.

The graphical representation of the cone shows the points taken as a red circle with a blue circle and a point joined by grey lines representing the best fit cone through those points. You can adjust the scale of the pictorial representation by holding the left mouse button down while, sliding the pointer either to the left or right to increase or decrease the scale respectively.

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. See chapter 13 - Erasing Data Points.

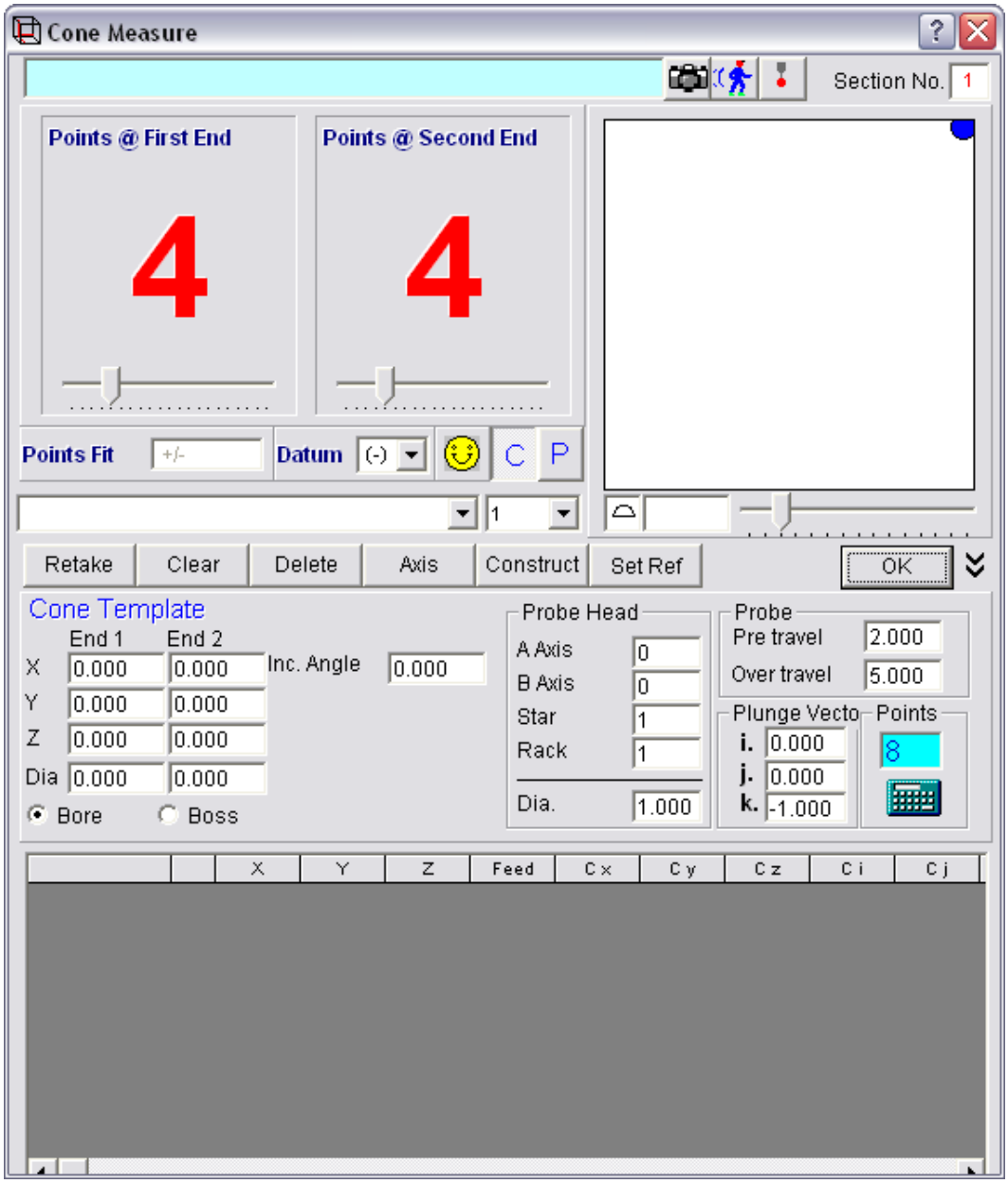
14.3 Off Line Programming

Instead of directly measuring a cone using the probe it is possible to enter the co-ordinates for the cone and let the software calculate the measurement points to be taken uniformly around it.

www.EngineeringBooksPdf.com

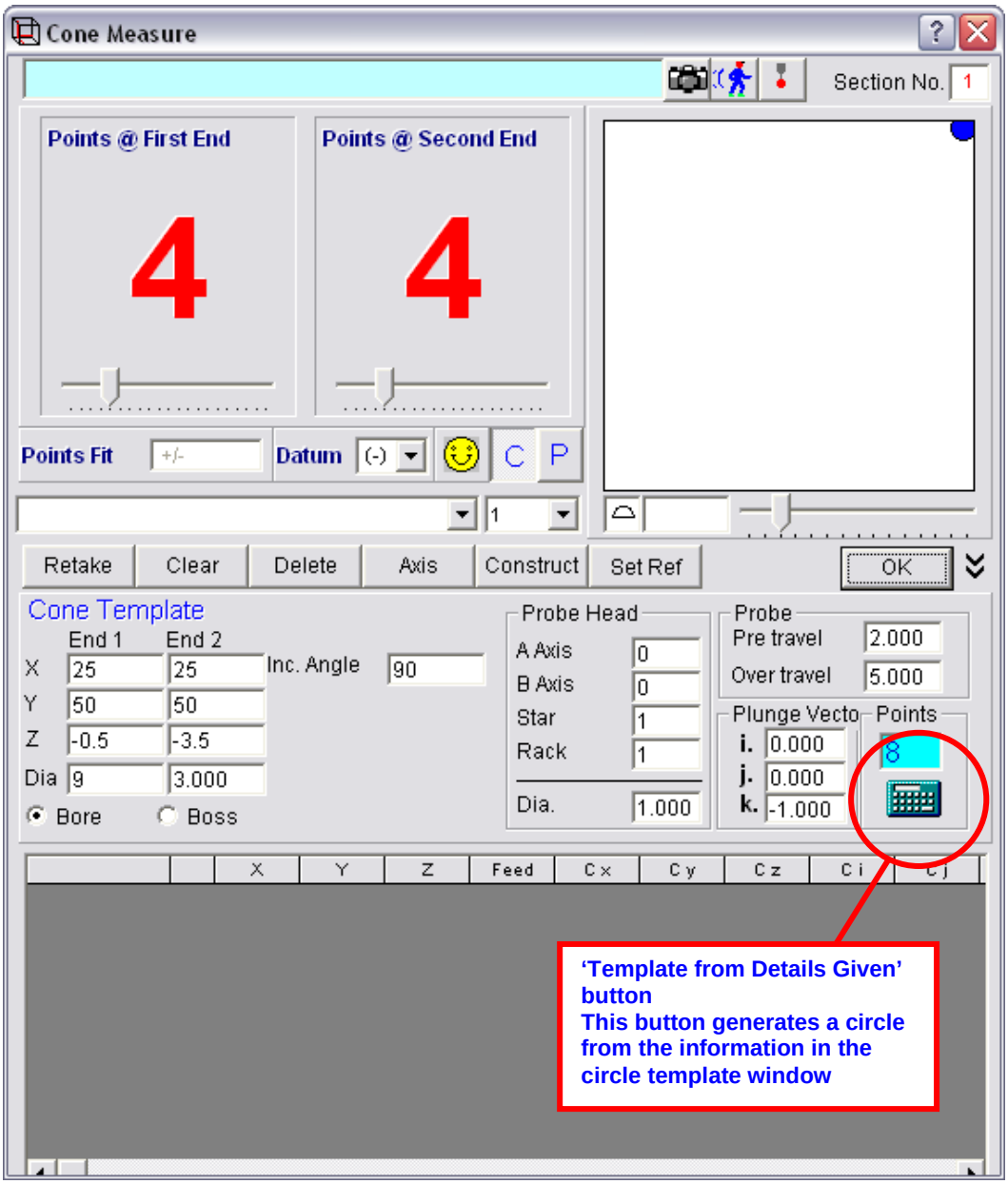
In this situation note that the points calculated will always be evenly distributed around the cone, even if there are other features or obstacles in the way:

To create an off-line program for a cone click on the Cone Measure button, from the main screen to bring up the Cone Measure window. Now instead of taking points with the probe we are going to type in the template details directly:



www.EngineeringBooksPdf.com

A cone is defined by entering the X,Y,Z co-ordinates of both ends of its centre line and its diameter at each end. Assume we have a 90 degree countersunk hole that we wish to measure as a cone at X=25, Y=50 in the Z reference plane (Z=0). The diameter of the cone at the top face is 10mm and we wish to use a 2mm stylus. Enter the co-ordinates for the cone into the Cone Template section of the window, as follows:



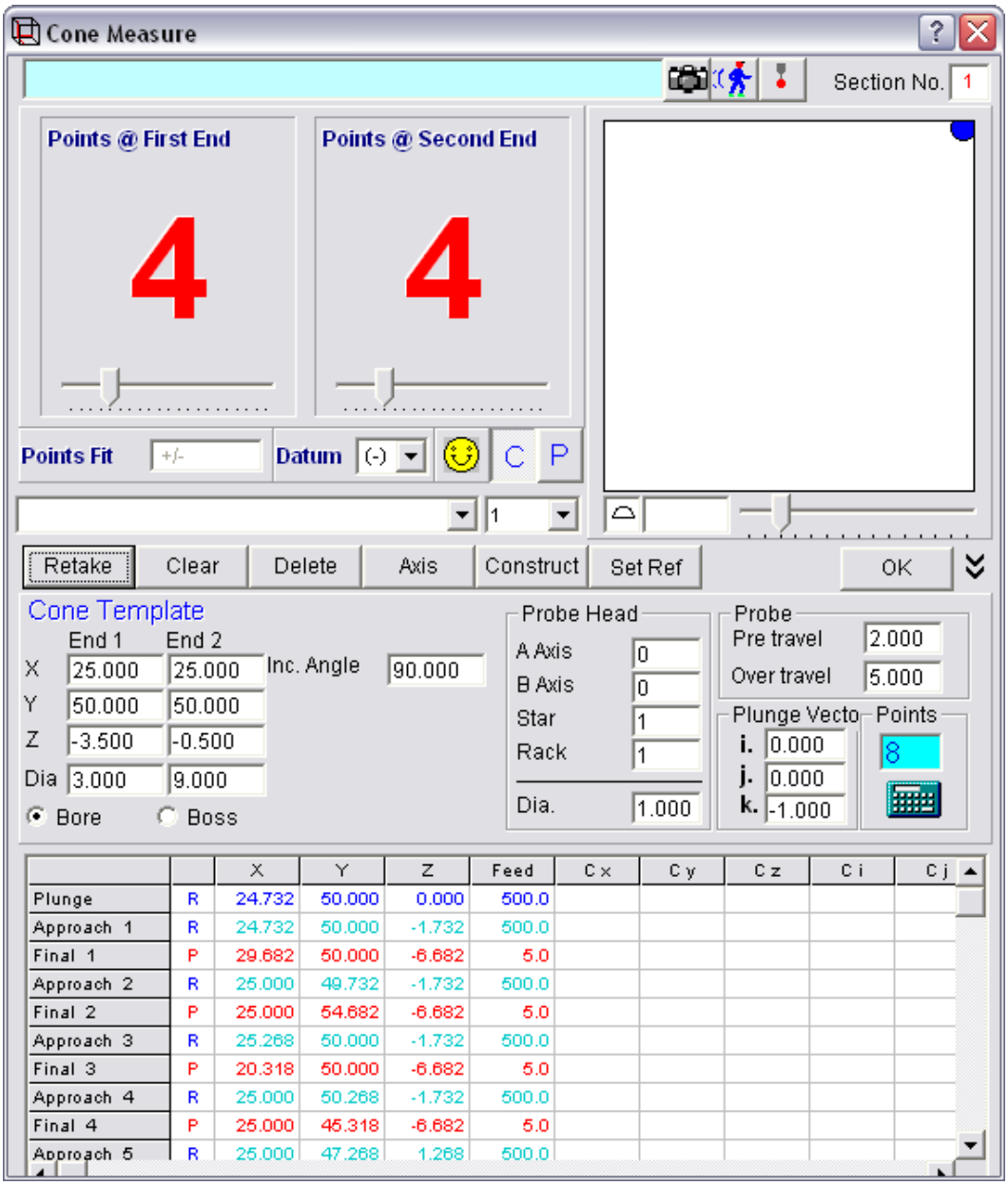
Make the top end of the cone at Z=-0.5 so that all points are taken with the centre of the stylus 0.5mm below the surface (you may need to increase this value if the hole has a larger chamfer, for instance). Make the bottom of the cone at Z=-3.5 as this is about as far as we can go down with a 2mm stylus. The diameter at the top end of the cone (0.5mm below the surface) is 9mm. Also the diameter at the bottom end of the defined cone (3.5mm below the surface) will be 3mm.

www.EngineeringBooksPdf.com

The pre-travel and the over travel will be set at their default values, but can be modified as required. Similarly you can change the probe head position and plunge vector (direction from which the probe will approach the cone) if required. Note that the default vector is defined by the orientation of the probe at the time of programming. The default number of points on the cone has been set to 8.

Note that if we enter the diameter of the cone at both ends then the include angle will be calculated automatically. Alternatively you can enter the diameter of the cone at end 1 and the include angle and the software will automatically calculate the diameter at end 2.

Now click on the 'Template from Details Given' button. The probe moves grid will now be automatically filled in, and the 8 points will be positioned at the around of the cone defined:

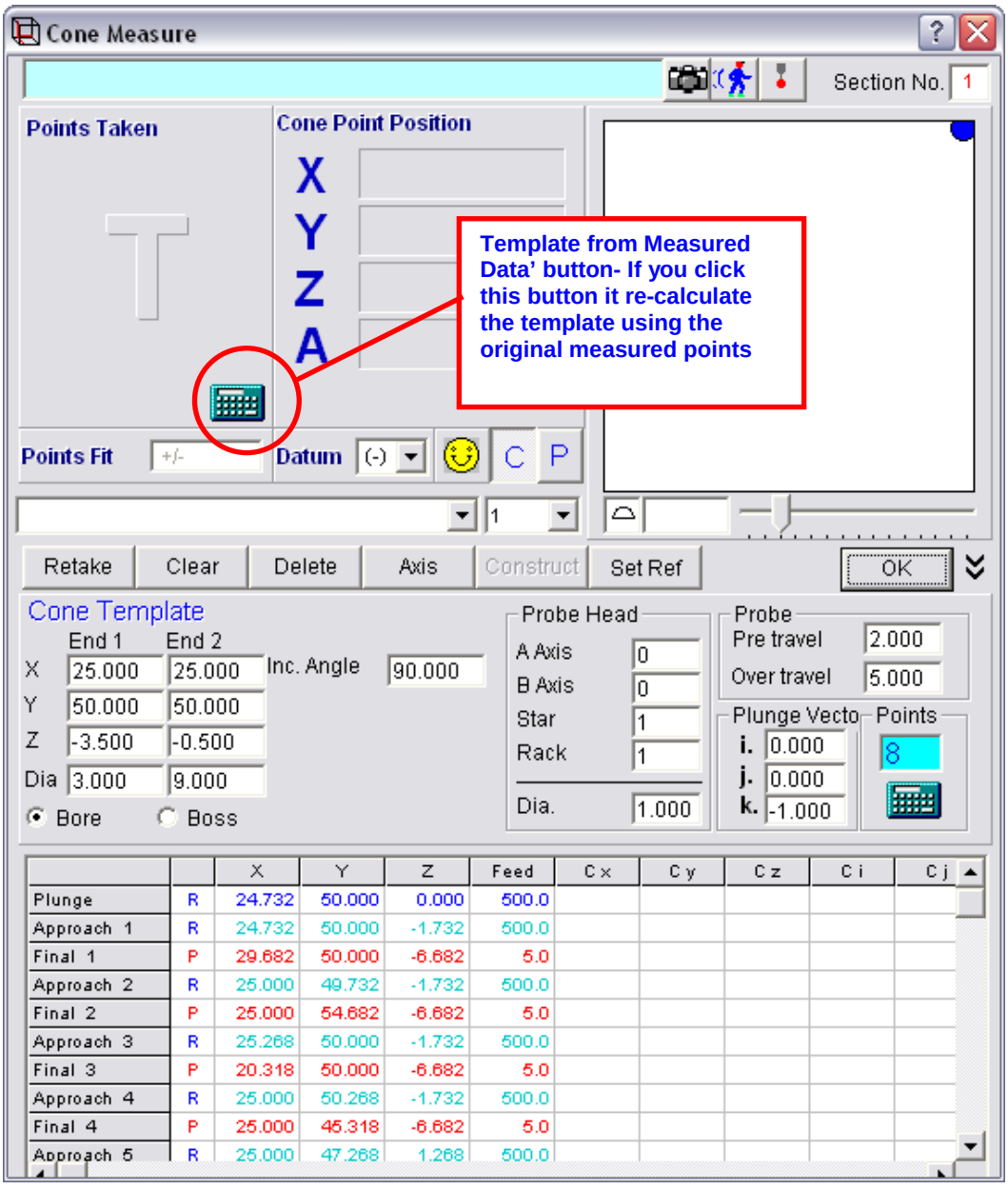


www.EngineeringBooksPdf.com

You can now click on OK (or Set Ref and OK) as desired. The Cone will appear as a grey template in the main screen.

14.4 The 'Template from Measured Data' button

When you create a program unit using the 'teach and repeat' method the program template is automatically calculated from measurement points taken.



www.EngineeringBooksPdf.com

If however, you need to re-calculate the template at any time, but cannot use the template details (for instance the points cannot be equi-spaced) then it is possible as follows:

- Clear the existing points, by clicking on the ‘Clear’ button.
- Take some new measurement points with the probe.
- Click on the ‘Template from Measured Data’ button to create a new program template from the points taken.

Note when you click on the ‘Template from Measured Data’ button the probe moves grid is re-calculated using the points taken. These points are shown in the top left hand corner of the window.
It is bad practice to re-calculate the template for a feature that has been set as a reference once the program has been run.

14.5 When to OK a Cone, and when to reference it.

Having measured a cone, you can either click on the ‘OK’ button or the ‘Set Ref ‘ button.

If you OK the cone, the measurement window will close, and the program will return to the main screen.
If any, or all of the graphics screens are blank, it is probably because they are outside the range currently shown on the screen. In this instance you will need to zoom to the appropriate position. See chapter 14 - Manipulating the Graphics.

The graphical representation of the cone should be shown as a solid blue circle representing the large end of the cone, with grey lines showing the shape and depth of the cone. This should be visible in all of the 3 views.

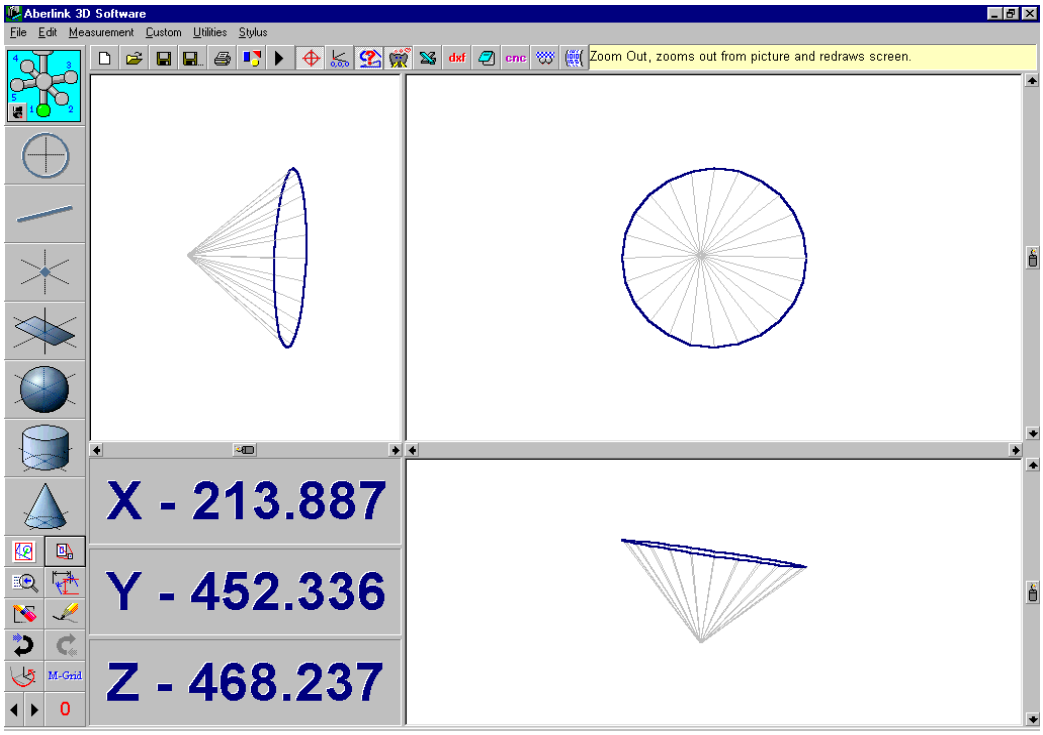


Fig. 14.5-1 The Main Screen showing a Cone that has been Ok’d

www.EngineeringBooksPdf.com

The angle at which the cone is shown will be relative to any previous references that have been set, or relative to the machine axes if no references have been set.

If you set the cone as a reference by clicking on 'Set Ref' then the measurement window will say open with the referencing doing two things. Firstly it will align the pictorial representation to the axis of the cone, and secondly it will set a point in space to X=0, Y=0, Z=0. This point, and also the alignment depends on whether a reference plane has previously been set.

If a plane has previously been set as a reference, the cone will be used to define 2 further degrees of freedom (see chapter 5 – Aligning the axes of a component), and will define one of the orthogonal planes to the reference plane.

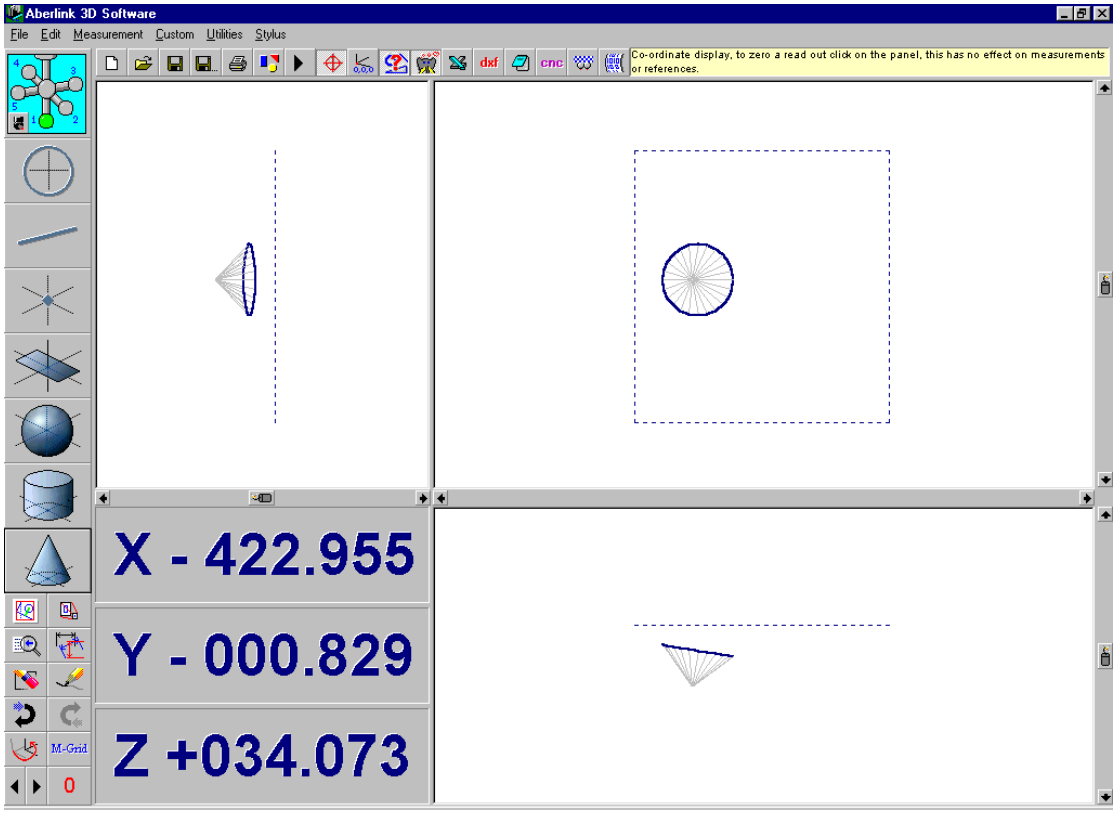
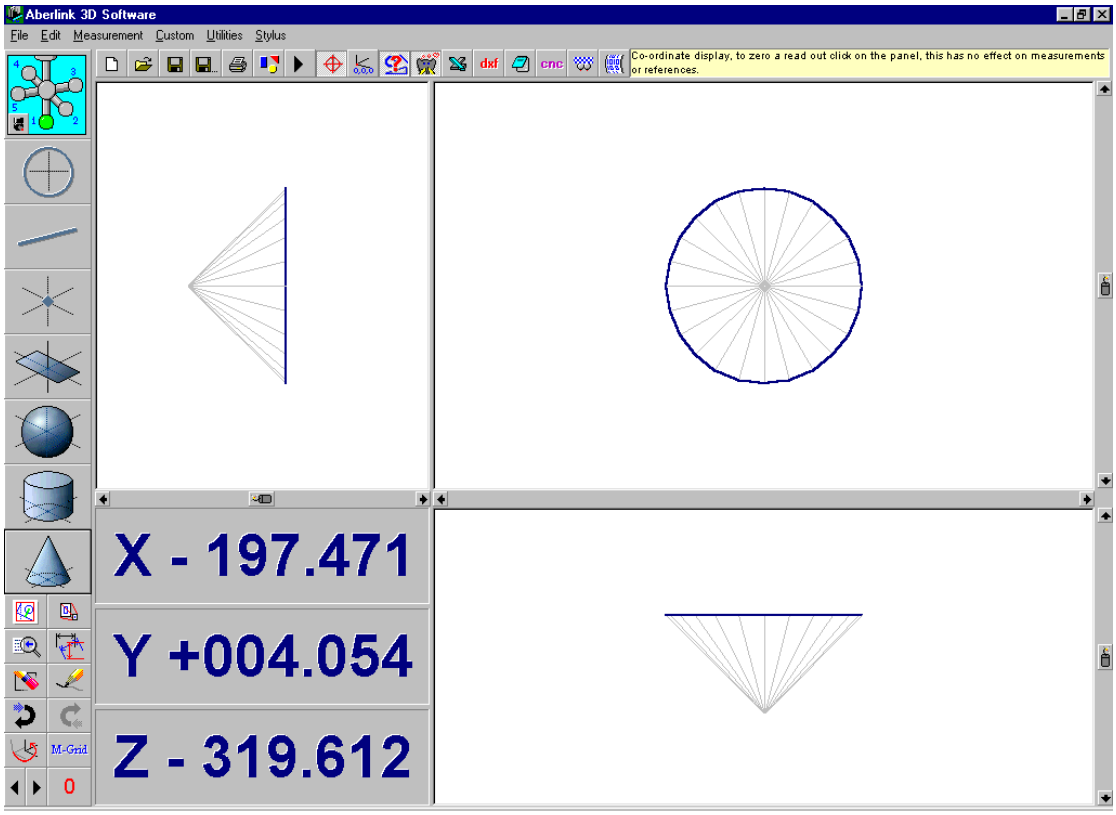


Fig. 14.5-2 A referenced cone with a reference plane

If however, no reference plane has previously been set, then the graphics picture will be aligned to the axis of the cone in all 3 views.

www.EngineeringBooksPdf.com



Note – Only one cone can ever be set as a reference at any point in time. If a second cone (or line or point) is set as a reference, this will override the initial reference, and the graphical representation will now re-align itself to the second cone.

14.6 Measuring the maximum diameter of a cone.

Often it will be important to know the diameter of the cone at its large end, where it meets a plane. Obviously the measured cone will only have information over the area that the actual measurement points were taken.

In this case the required diameter can be obtained by constructing a circle at the intersection of the cone and the plane. This is described in section 8.2 – Constructing Circles.

www.EngineeringBooksPdf.com

14.7 Determining the Position on a cone of a given diameter.

Sometimes it may be necessary to determine at what position on a cone is a given diameter. This is done by constructing a circle (as described in chapter 8.2) and selecting the cone twice. This will bring up the following prompt:

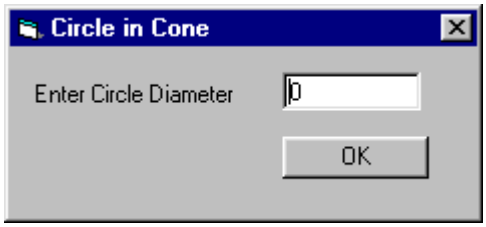


Fig. 14.5-1 The prompt for constructing a circle of given diameter on a cone

Now simply enter the desired diameter via the keyboard, and then click ‘OK’, and a circle will be constructed on the cone at this diameter.

www.EngineeringBooksPdf.com

15.0 THE CURVE MEASURE UNIT

The curve measure unit is a method for taking multiple point data over a surface without pre-programming each point.

Using 3 points to define an area and defining the required step size will allow the software to automatically scan discrete points over the surface.

The curve measure unit can therefore be used to take point data over a complex surface. However, to analyse this data or to fit curves or surfaces through the points the measurement file must be exported to CAD.

15.1 Creating a Curve Measure Program Unit

Background
The curve measure unit will scan discrete points over a surface, even of complex geometry. The software will intelligently predict the position of the next point on its scan path having determined the gradient from previous points measured.

The curve measure function can also cope with quite rapid changes in gradient or even step changes providing that the pre-travel has been set sufficiently high to cope with the distances involved.

The Aberlink 3D software will even compensate for the stylus radius so that the measured data truly represents the surface being inspected.

Analysis of the data must however be performed outside of the Aberlink 3D software by exporting the measurement file either to CAD or to another package perhaps using the Excel function.

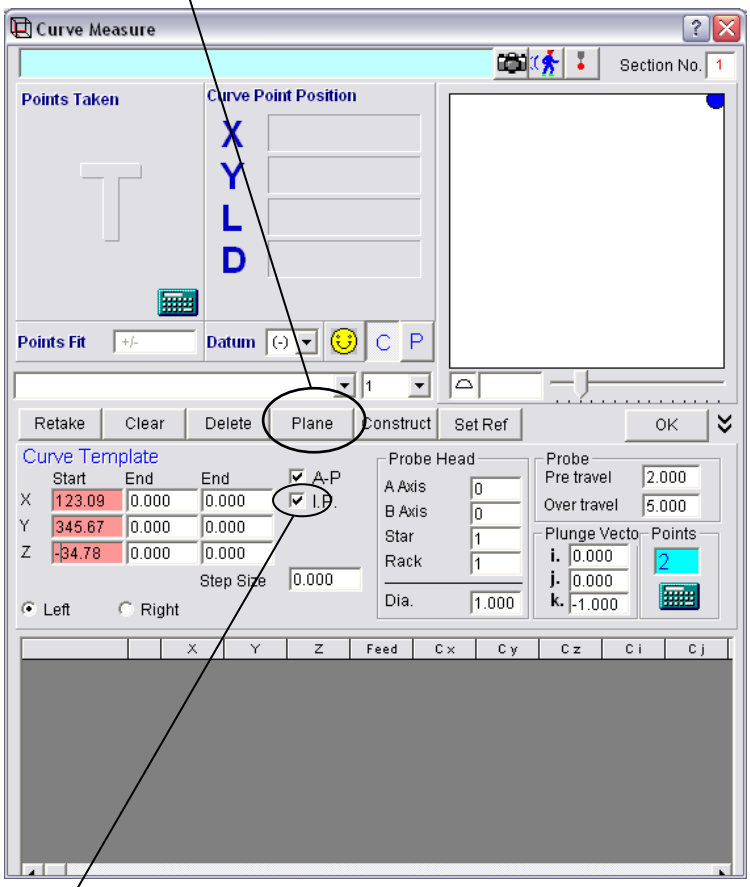
15.2 Measuring a 2D curves in a fixed plane (XY,ZX,YZ) using the curve button.

To create a 2 D curve in a fixed plane , click on the Curve Measure button, from the main screen:

This will bring up the Curve Measure window:

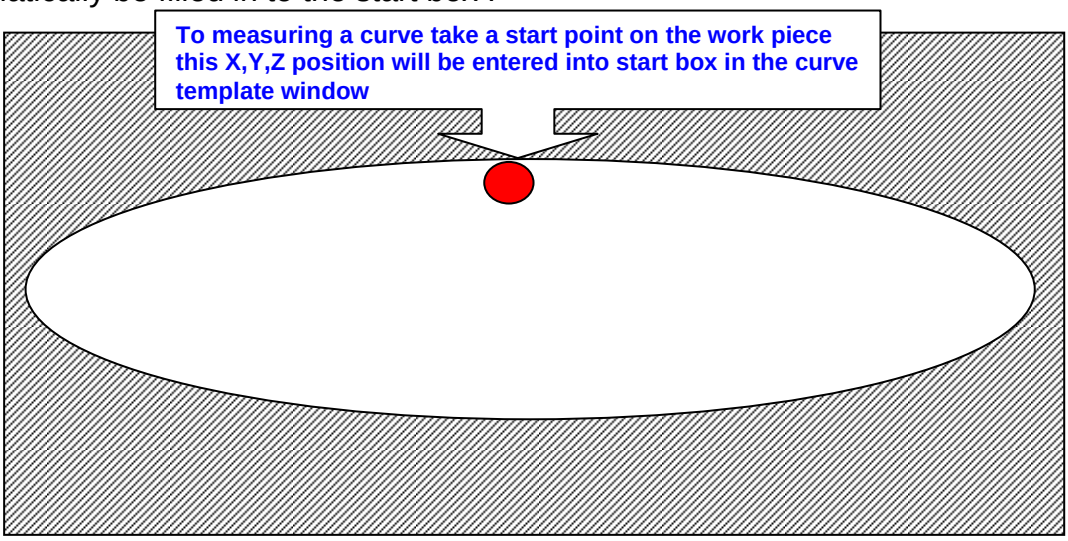
www.EngineeringBooksPdf.com

1. Click on the plane button to select which plane you wish to measure 2D curve in(XY,ZX,YZ)

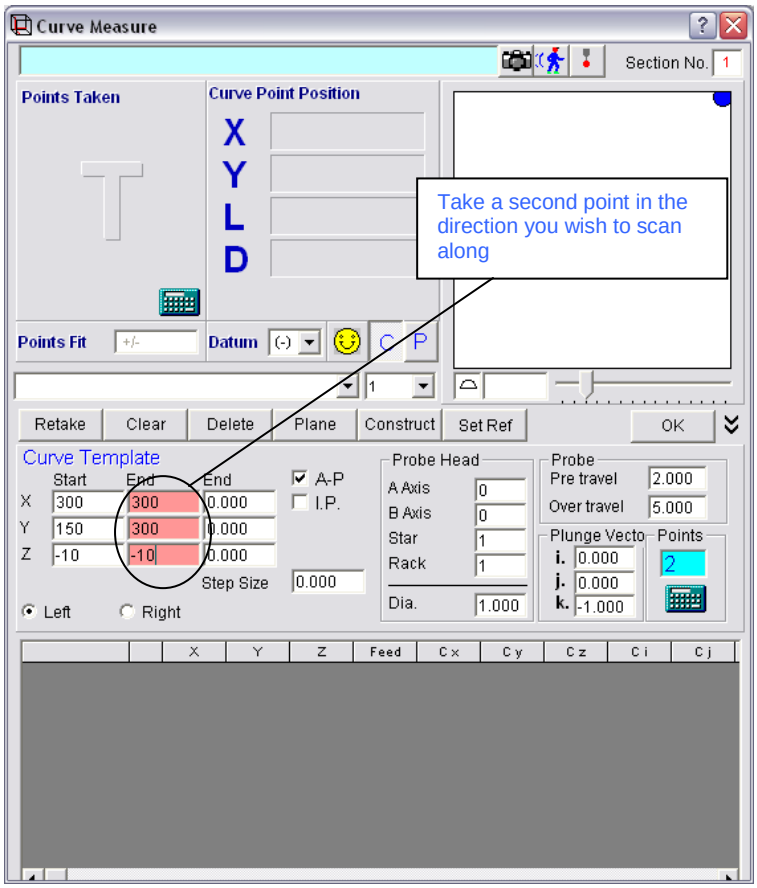


2. Tick the 'I.P'(In Plane) box

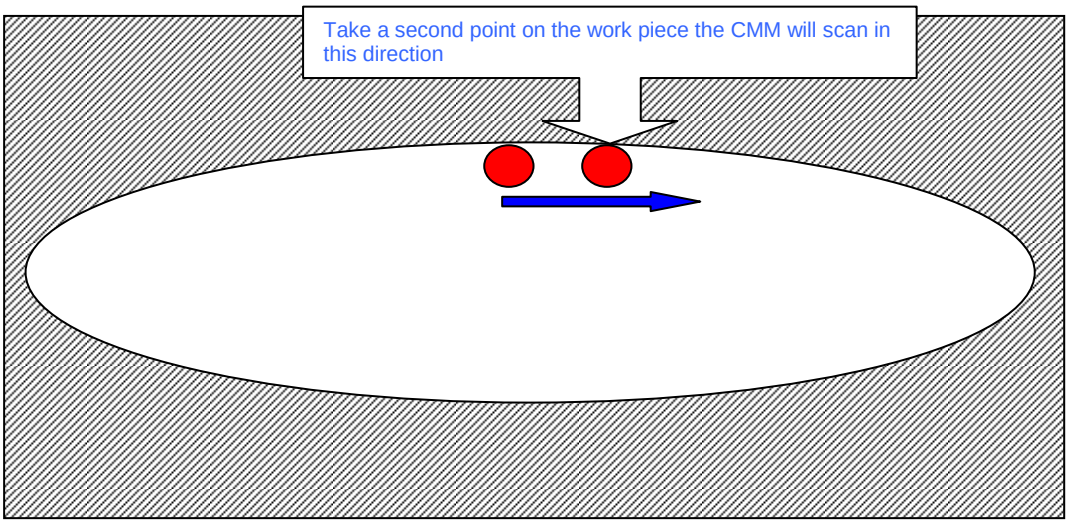
The cursor will already be positioned in the boxes under the heading 'Start' and the colour of the boxes will turn red. Take the first point at the beginning of the 2D curve you wish to scan and the X,Y,Z position of this point will automatically be filled in to the start box :



www.EngineeringBooksPdf.com

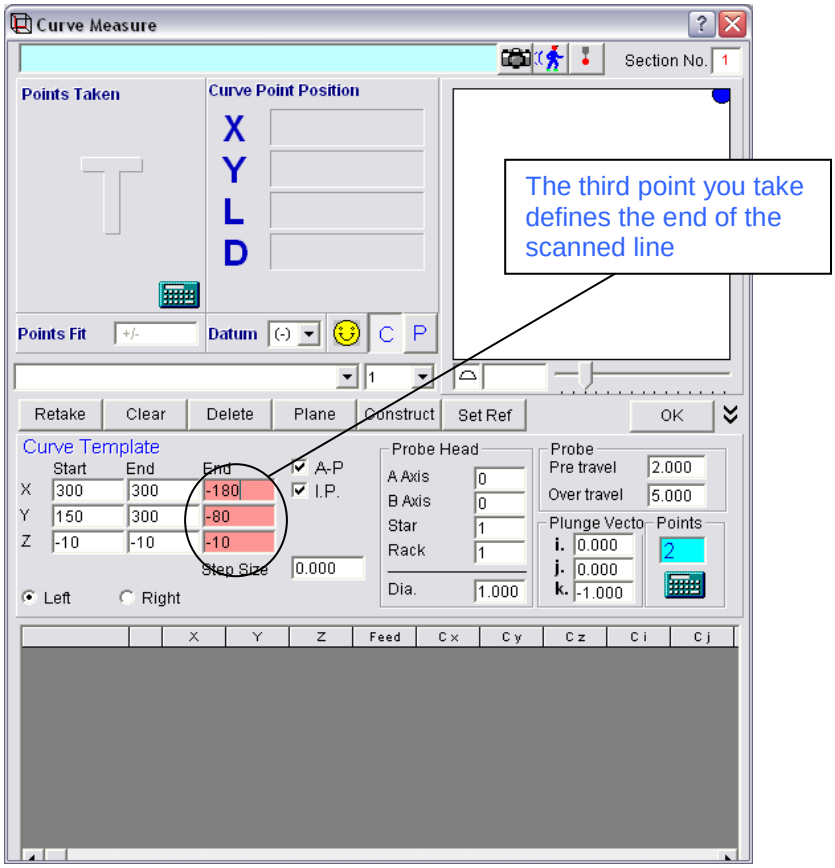


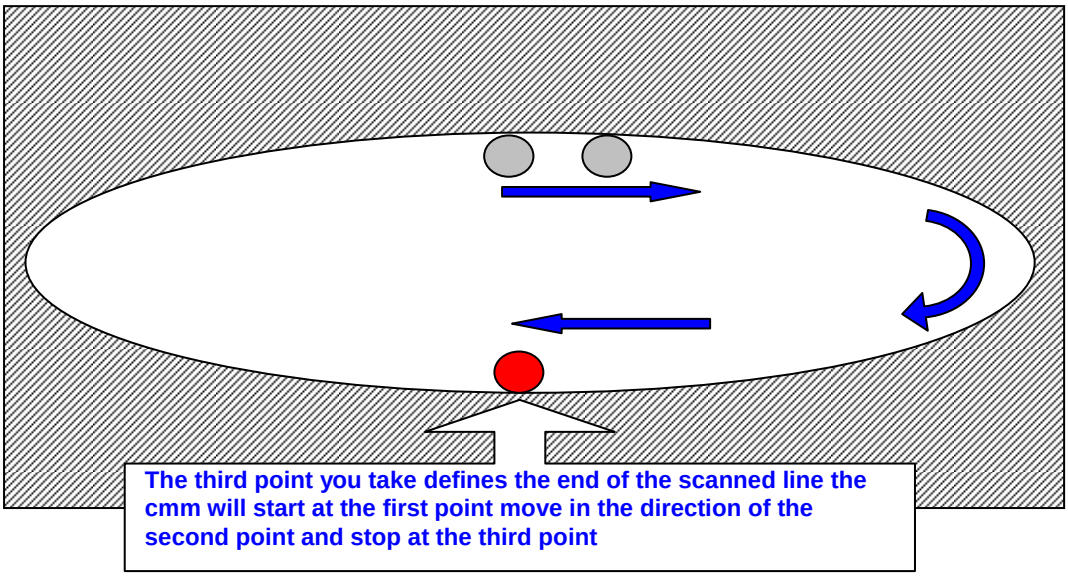
Also the cursor will automatically jump to the next set of boxes under the heading ‘End 1’. Now take a second point to defining the direction wish to scan along. Again the X,Y,Z position of this point will automatically be filled in and the cursor jump to the last set of boxes labelled ‘End’.



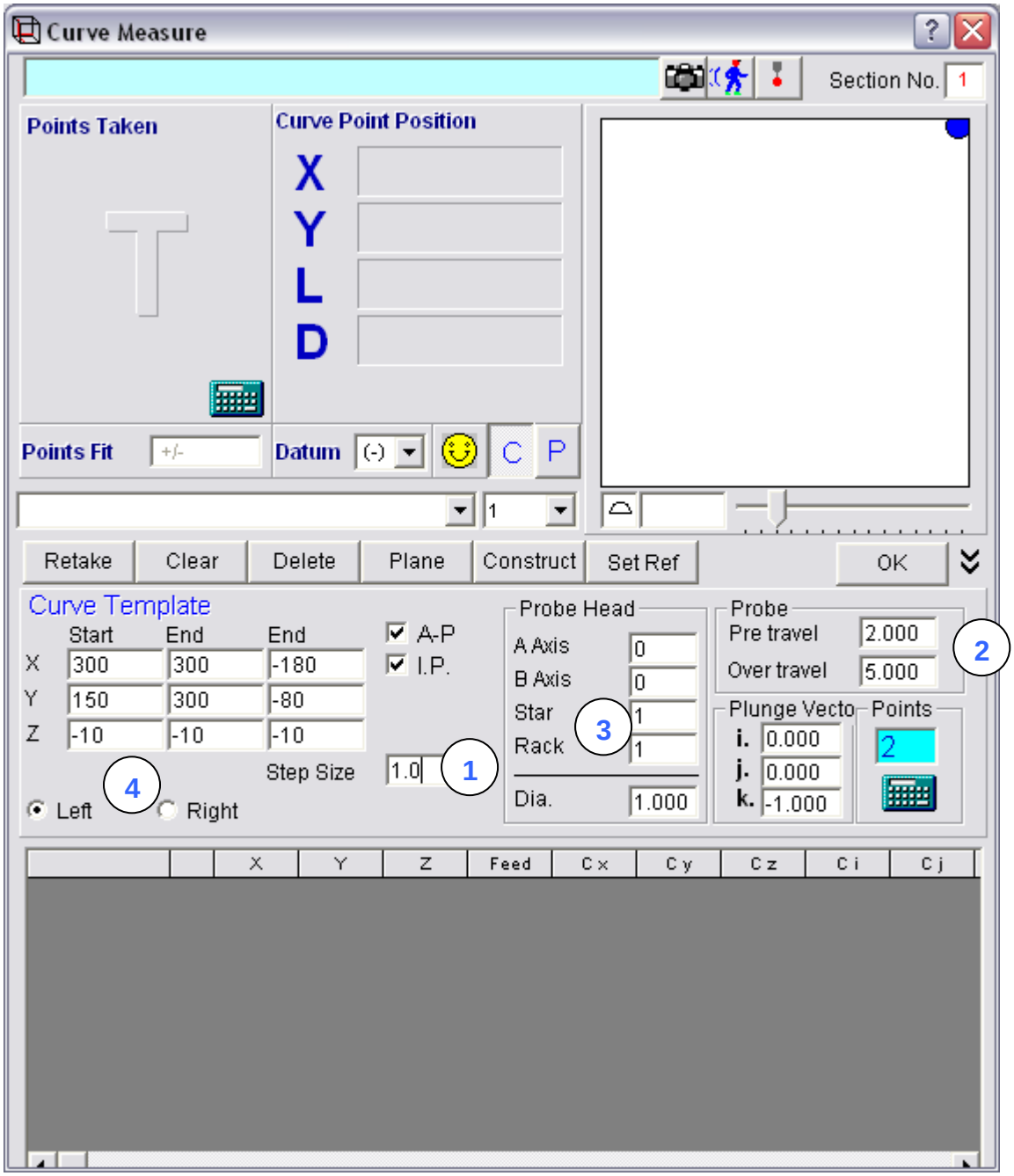
Now take a third point defining the end of the curve. All three sets of X,Y,Z position boxes should now be filled in defining the length and direction of the curve you wish to scan.

www.EngineeringBooksPdf.com





www.EngineeringBooksPdf.com



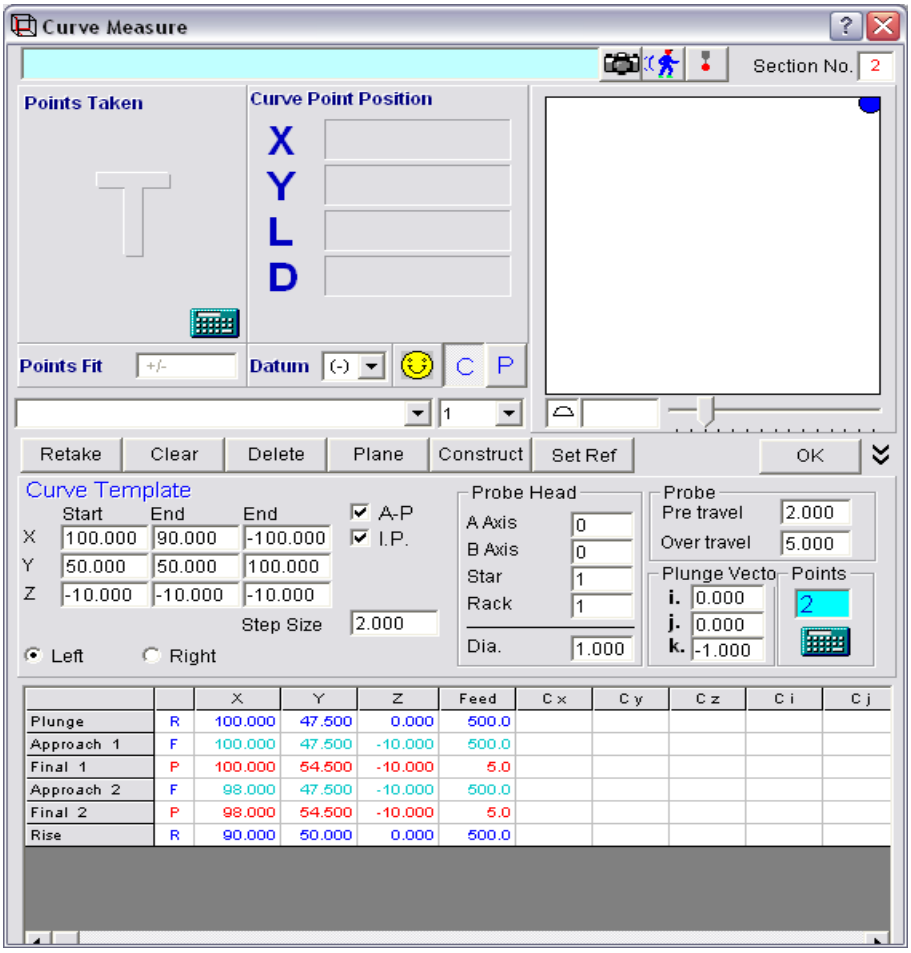
1. You will now need to enter the step size desired. For instance if you choose 2 (mm) the area will divided into a grid and points will be taken at 2mm intervals

2. The Pre-travel and over-travel boxes will be filled in with the default values. Make sure that these are suitable for the surface that you are trying to measure. In general the more severe the gradients that you are trying to measure the larger values you will require, but the slower the program will run.

3. Also make sure that the probe head position (if relevant) and approach vector are as required.

www.EngineeringBooksPdf.com

4. The last thing to check is the ‘Left’ and ‘Right’ buttons. These define which side of the defined surface will be measured. It may be easier to determine which button is required after the probe moves have been calculated once you click on the ‘Template from Details Given’ button. For example assume that we are measuring a surface primarily in the XY plane, and we wish to probe down onto the surface:



The button defaults to ‘Left’ – leave it on left and click on the ‘Template from Details Given’(calculator) button. The probe moves grid will now be filled in with just 2 moves:

NOTE:-

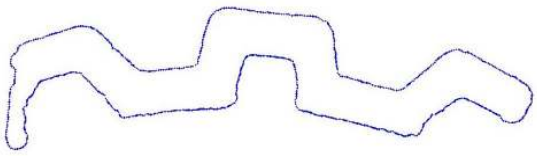
WHEN AUTOMATICALLY RUNNING THE CURVE MEASURE UNIT THE DRY RUN FACILITY WILL NOT BE OF ANY USE AND THEREFORE SHOULD BE SWITCHED OFF

15.3 Compare a Curve Profile to nominal DXF Profile

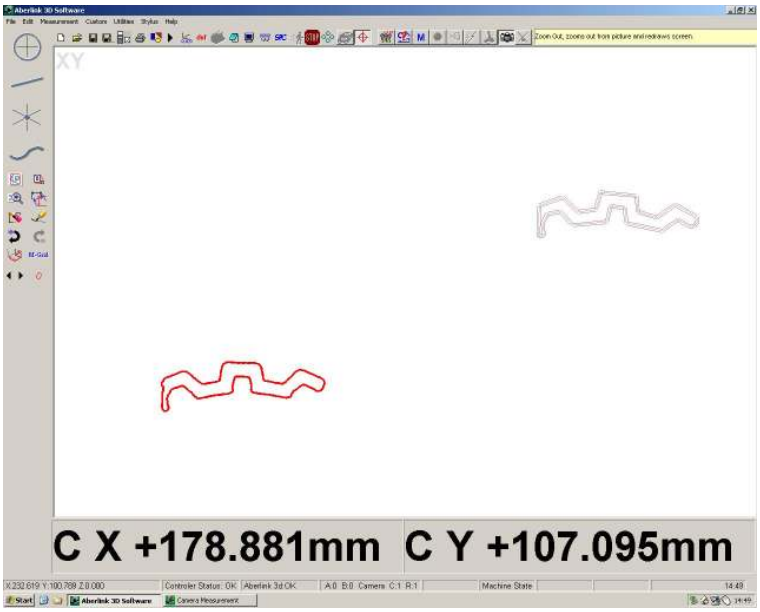
If the component shape has been measured as a Curve profile, it is possible to import the DXF file for the shape, and perform a best-fit optimisation to get the measured points to fit the DXF file.

www.EngineeringBooksPdf.com

Firstly measure the component as a Curve. This will appear as a series of blue dots in the Main Screen:



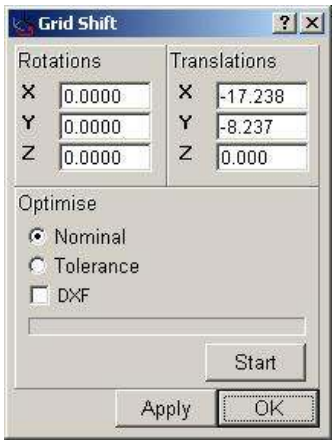
Select ‘Import As Curve Profile’ in the DXF window. Also type in the default profile tolerance required, and then browse to the DXF file. When you OK the DXF window the DXF file will be drawn in the Main Screen, but will not necessarily be in the same position as measured component, so it may be necessary to zoom out in order to see it:



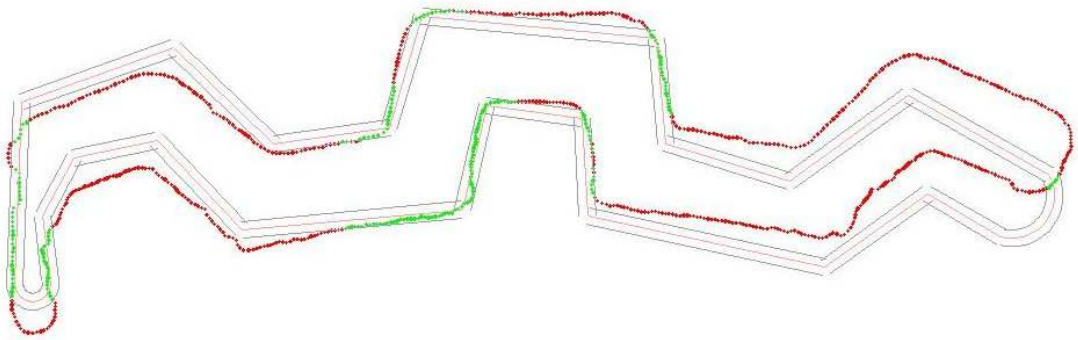
The nominal DXF file is shown as a red line, with the default tolerance shown as grey lines on either side.

www.EngineeringBooksPdf.com

Now open the Grid Shift window as follows:



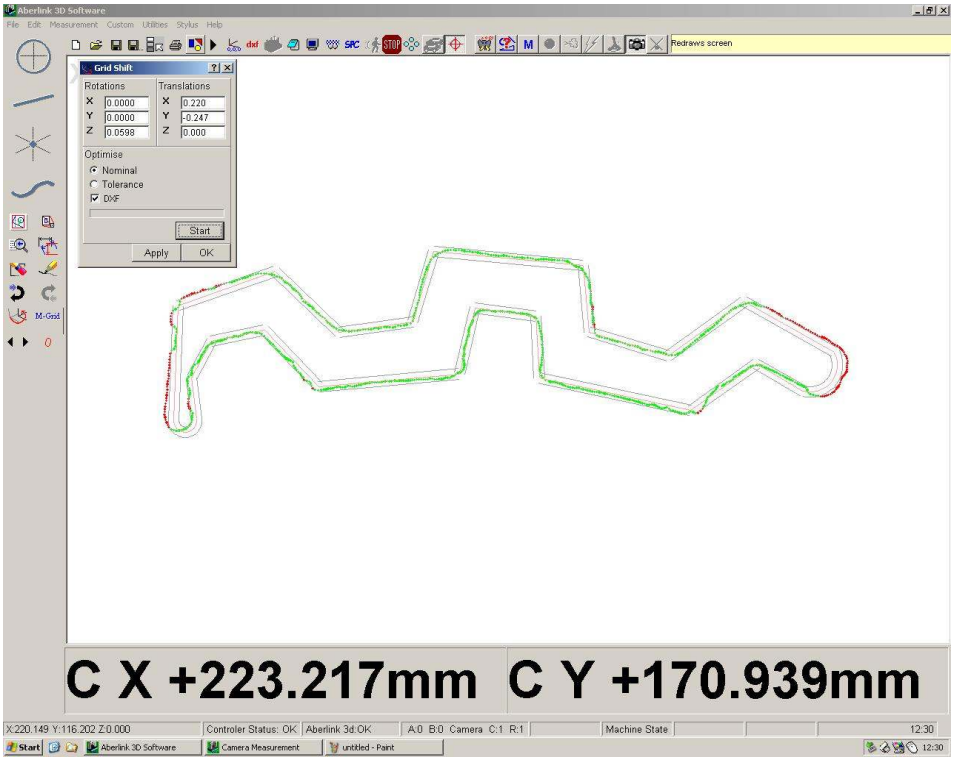
Whilst this window is open, the measured data points can be dragged using the right mouse button so that they are approximately over the top of the imported DXF file:



You will see that where the points are within the specified tolerance band they are shown in green, and where they are outside they are shown in red.

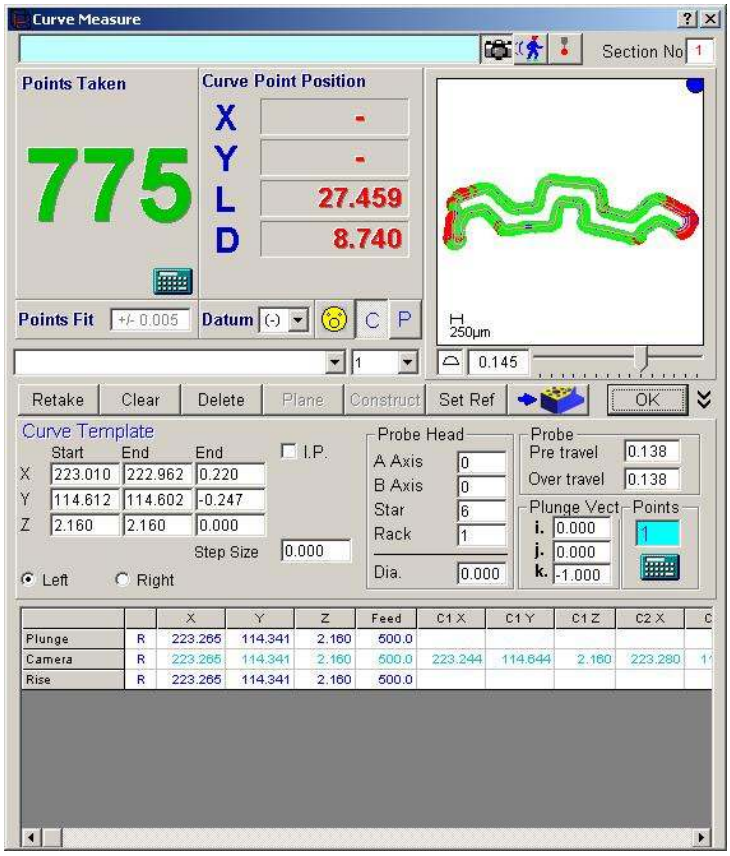
Now to perform a best fit, tick the DXF box under the ‘Optimise’ option and then click start. The software will now take a few seconds to calculate the rotation and translations required to fit the measured data to the DXF file. When it is finished the measured points will snap to the DXF file. Click OK and the software will return to the Main Screen:

www.EngineeringBooksPdf.com



At a glance the areas outside of the tolerance can be seen as they are shown in red. Right clicking on the representation of the curve (or using the Feature Select buttons) will re-open the Curve Measure window:

www.EngineeringBooksPdf.com



Right click in the graphic area and choose ‘expand’ to make the graphical representation fill the whole screen:

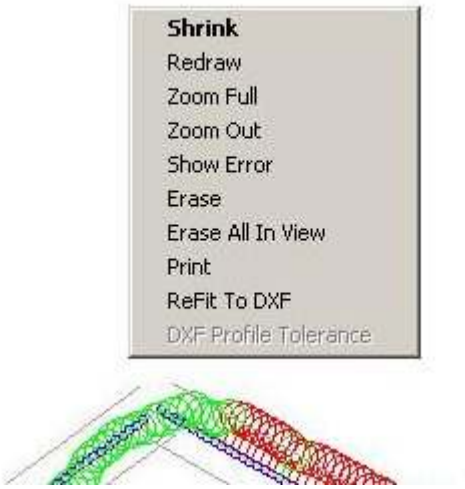
The measurement point shown in blue denotes the biggest error.

www.EngineeringBooksPdf.com

This value will be shown as the Profile of a Surface value in the bottom, left hand corner of the window.

The scale can be modified by using the slider bar, which is located next to the Profile of a Surface box. A scale indicator line is shown above the slider bar.

Right clicking in the window will bring up the following menu:



Shrink - will return to the full Curve Measure window.

Redraw - will refresh the image.

Zoom Full - will zoom to fit the whole of the Curve to fit the window.

Zoom Out - will zoom out by a fixed percentage.

Show Error - will bring up a box showing the error of any point right clicked on.

Erase - will erase any point right clicked on.

Erase All In View - will allow a box to be dragged over multiple points to be erased.

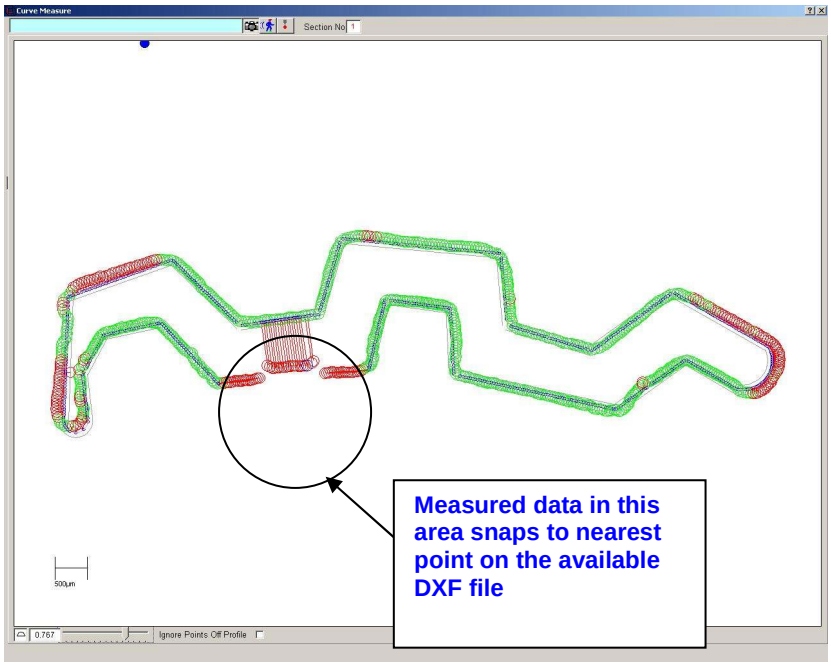
Print - will allow this representation to be printed as a Feature Profile report.

ReFit to DXF - will allow the best fit routine to be repeated.

DXF Profile Tolerance - will allow the default tolerance that was imported with the DXF file to be changed for any feature within the DXF file. Right click on that feature and select this option.

www.EngineeringBooksPdf.com

If the DXF file happens to contain gaps for any reason, this can produce errors because the error for the measured data within the gap(s) will be reported to the closest point on the DXF file, as follows:



In this situation simply tick the 'Ignore Points Off Profile' box, which will now produce a representation as follows:

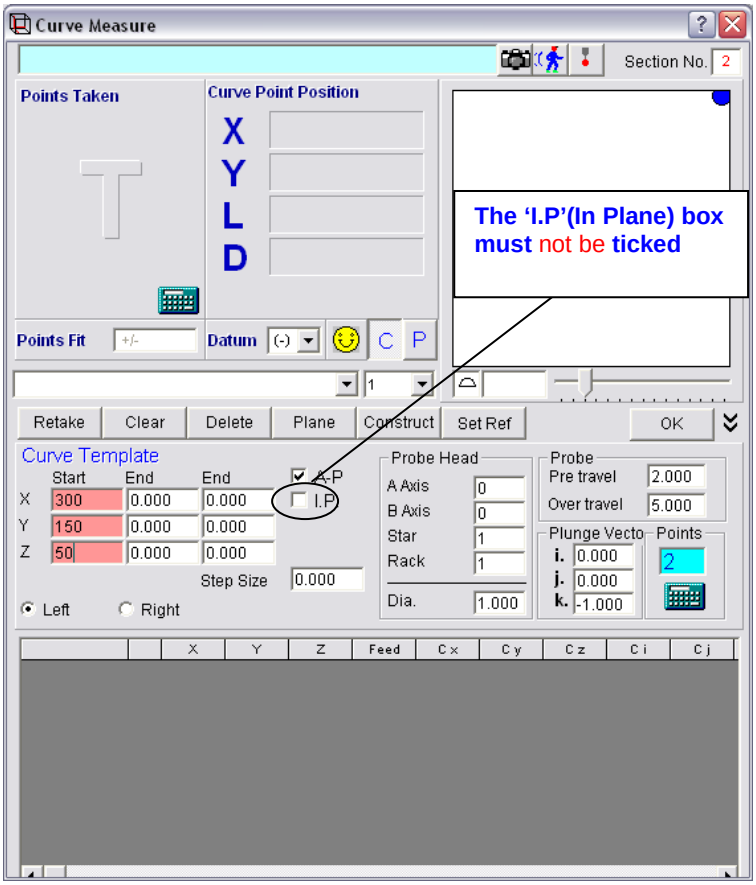
15.4 Measuring a 3D surface area using the curve button

By use 3 points to define an area the software will automatically scan the surface in a grid Pattern
To create a 3 D area, click on the Curve Measure button, from the main screen:

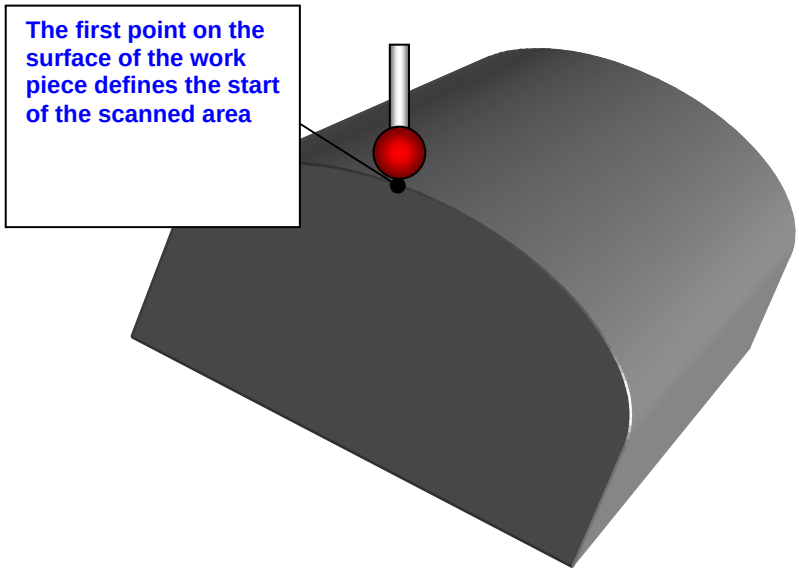
www.EngineeringBooksPdf.com

This will bring up the Curve Measure window:

The ‘I.P’(In Plane) box must **not be** ticked

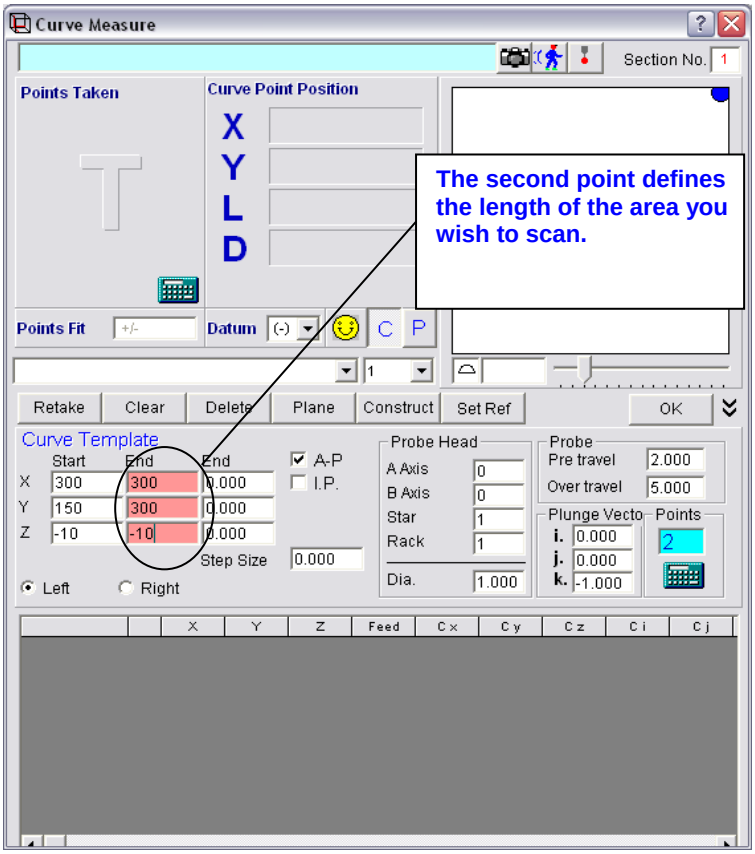


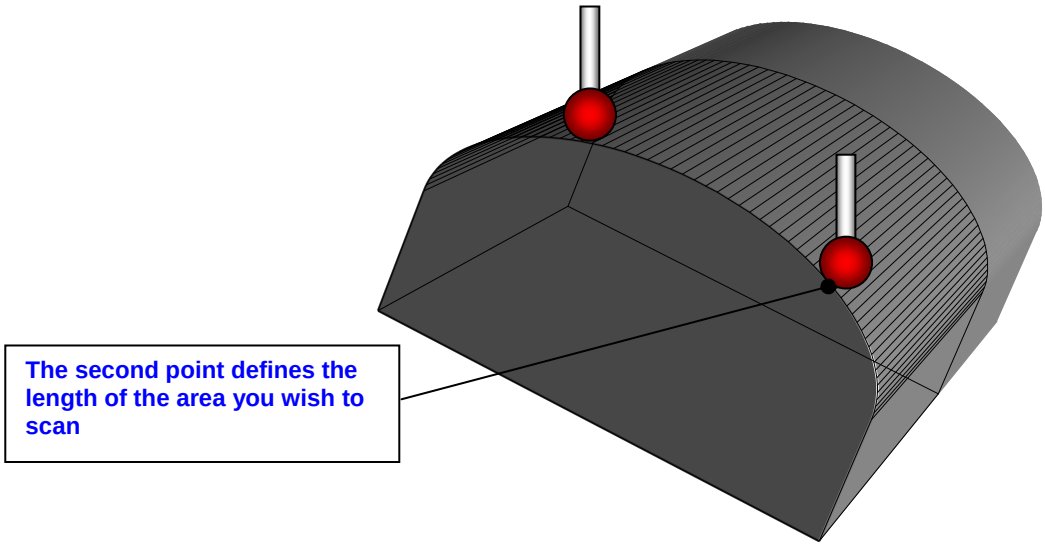
The cursor will already be positioned in the boxes under the heading ‘Start’ and the colour of the boxes will turn red. Take the first point at the beginning of the 3D area you wish to scan and the X,Y,Z position of this point will automatically be filled in to the start box :



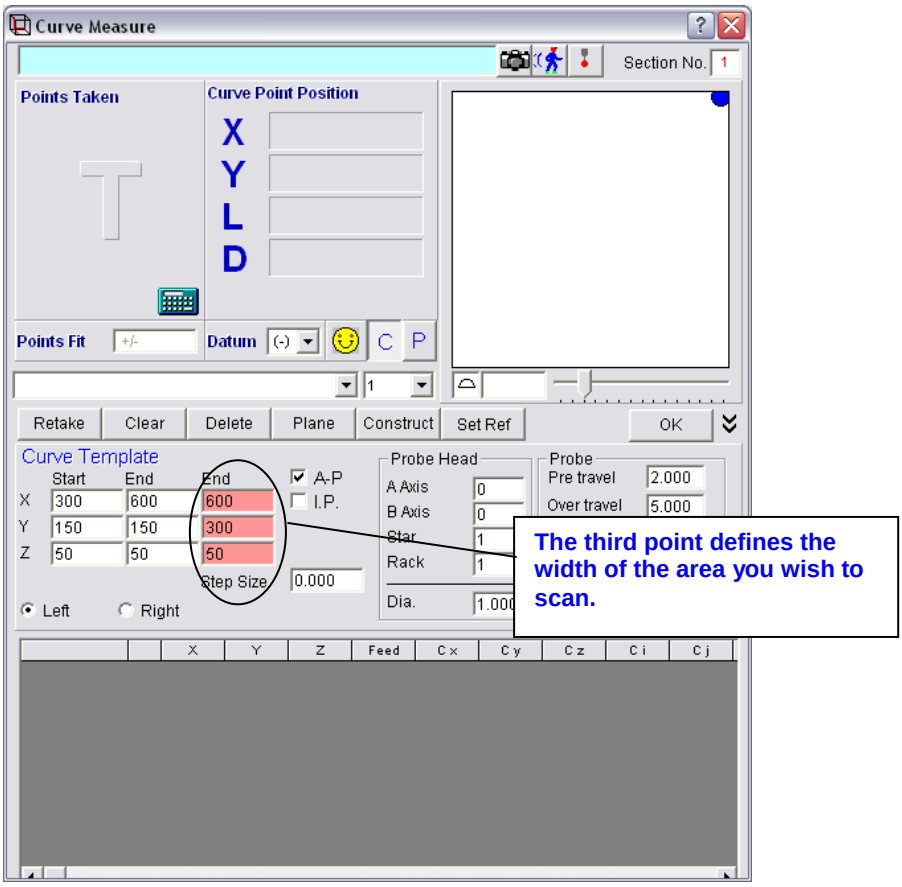
www.EngineeringBooksPdf.com

Also the cursor will automatically jump to the next set of boxes under the heading ‘End 1’. Now take a second point to defining the length of the area you wish to scan. Again the X,Y,Z position of this point will automatically be filled in and the cursor jump to the last set of boxes labelled ‘End’

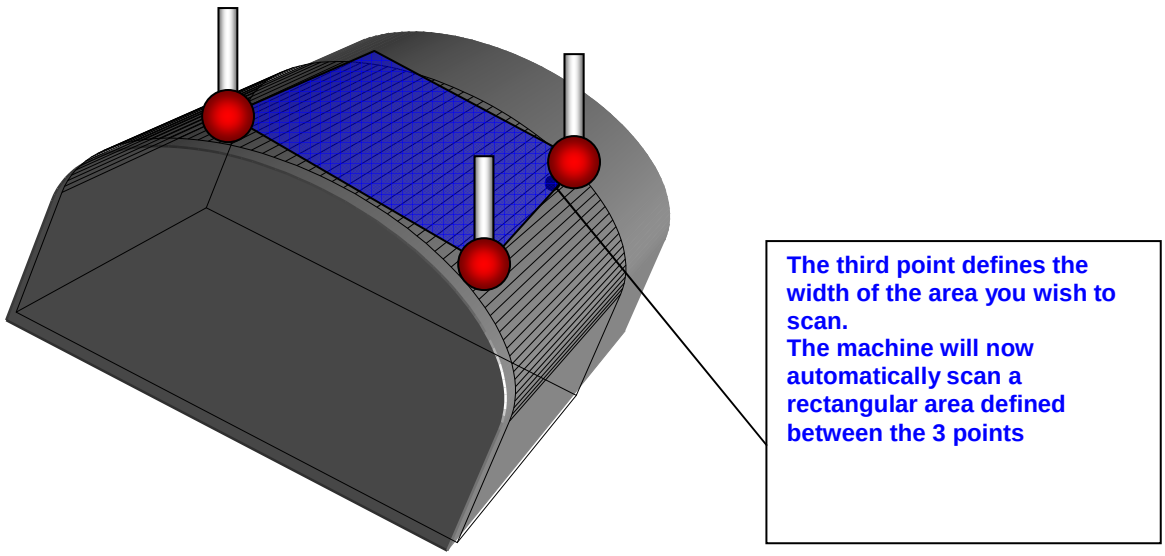




www.EngineeringBooksPdf.com



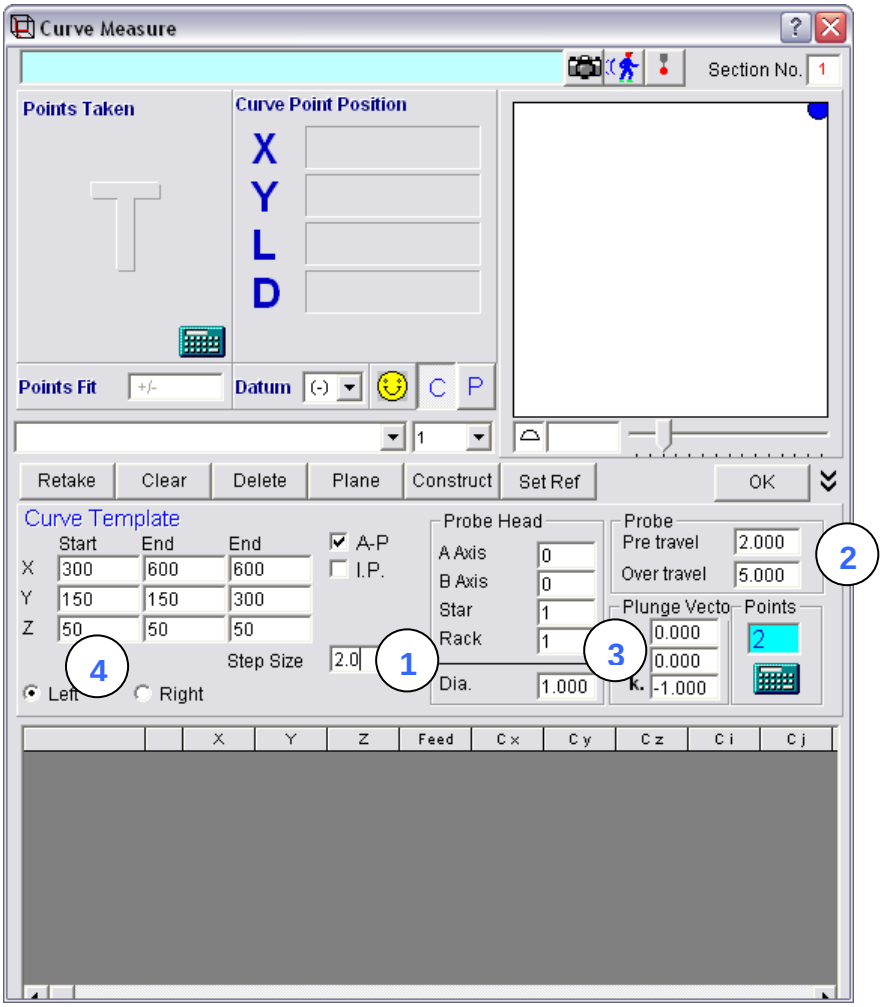
Now take a third point defining the width. All three sets of X,Y,Z position boxes should now be filled in defining the length and width of the area you wish to scan.



www.EngineeringBooksPdf.com

1. You will now need to enter the step size desired. For instance if you choose 2 (mm) the area will divided into a grid and points will be taken at 2mm intervals

2. The Pre-travel and over-travel boxes will be filled in with the default values. Make sure that these are suitable for the surface that you are trying to measure. In general the more severe the gradients that you are trying to measure the larger values you will require, but the slower the program will run.

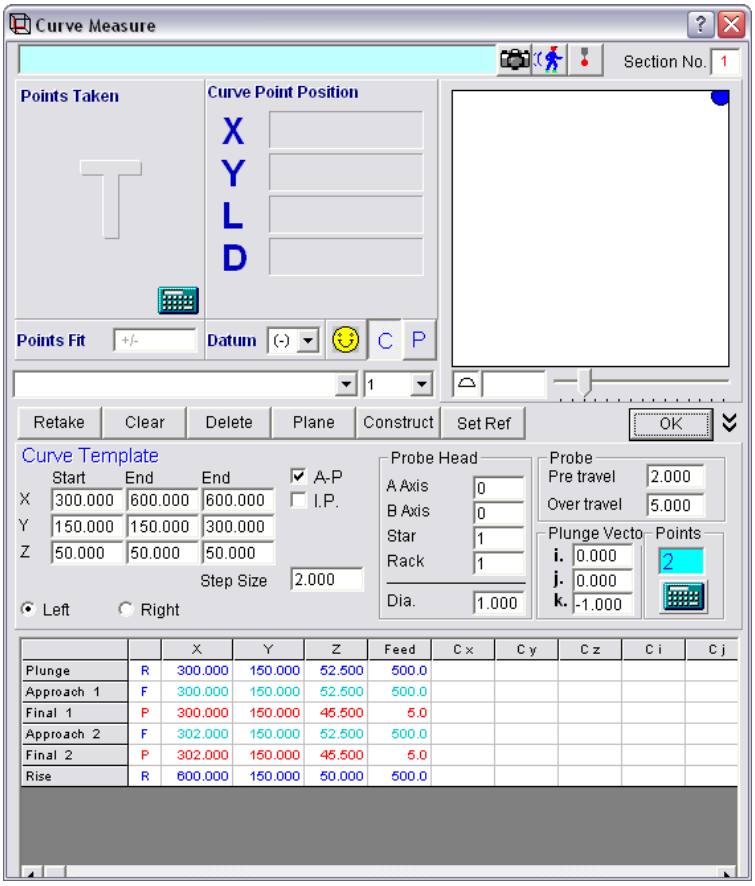


3. Also make sure that the probe head position (if relevant) and approach vector are as required.

4. The last thing to check is the ‘Left’ and ‘Right’ buttons. These define which side of the defined surface will be measured. It may be easier to determine which button is required after the probe moves have been calculated once you click on the ‘Template from Details Given’ button. For example assume that we are measuring a surface primarily in the XY plane, and we wish to probe down onto the surface:

www.EngineeringBooksPdf.com

The button defaults to ‘Left’ – leave it on left and click on the ‘Template from Details Given’(calculator) button. The probe moves grid will now be filled in with just 2 moves:



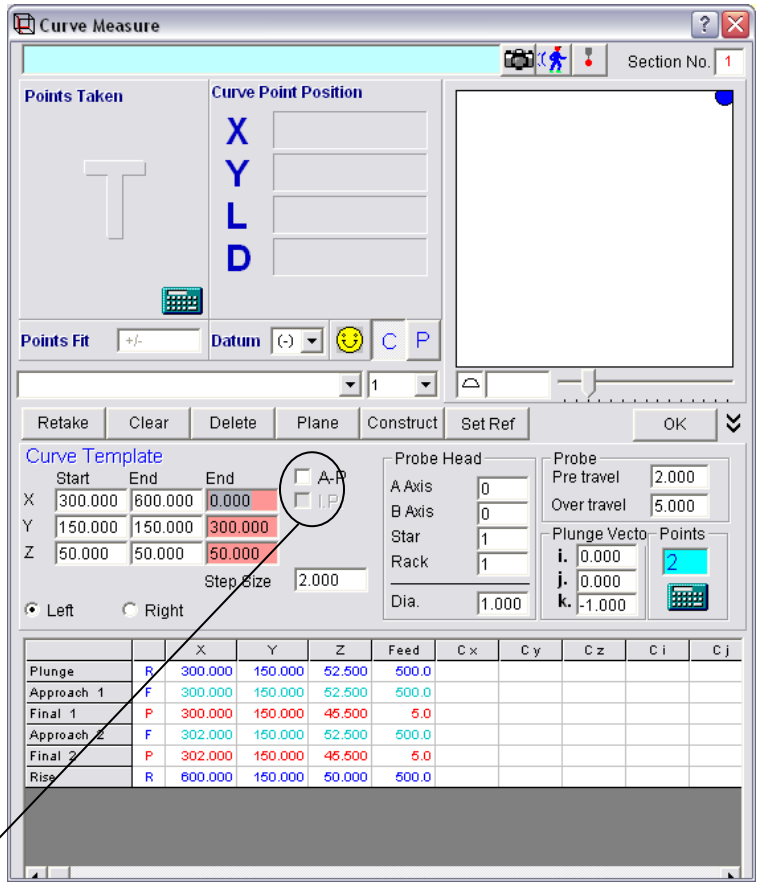
NOTE:-

When automatically running the Curve Measure Unit the dry run facility will not be of any use and therefore should be switched off

www.EngineeringBooksPdf.com

15.5 Measuring multiple points using the curve button

This allows you measure multiple points on a work piece as one feature, this function is used a lot when measuring against a CAD model.

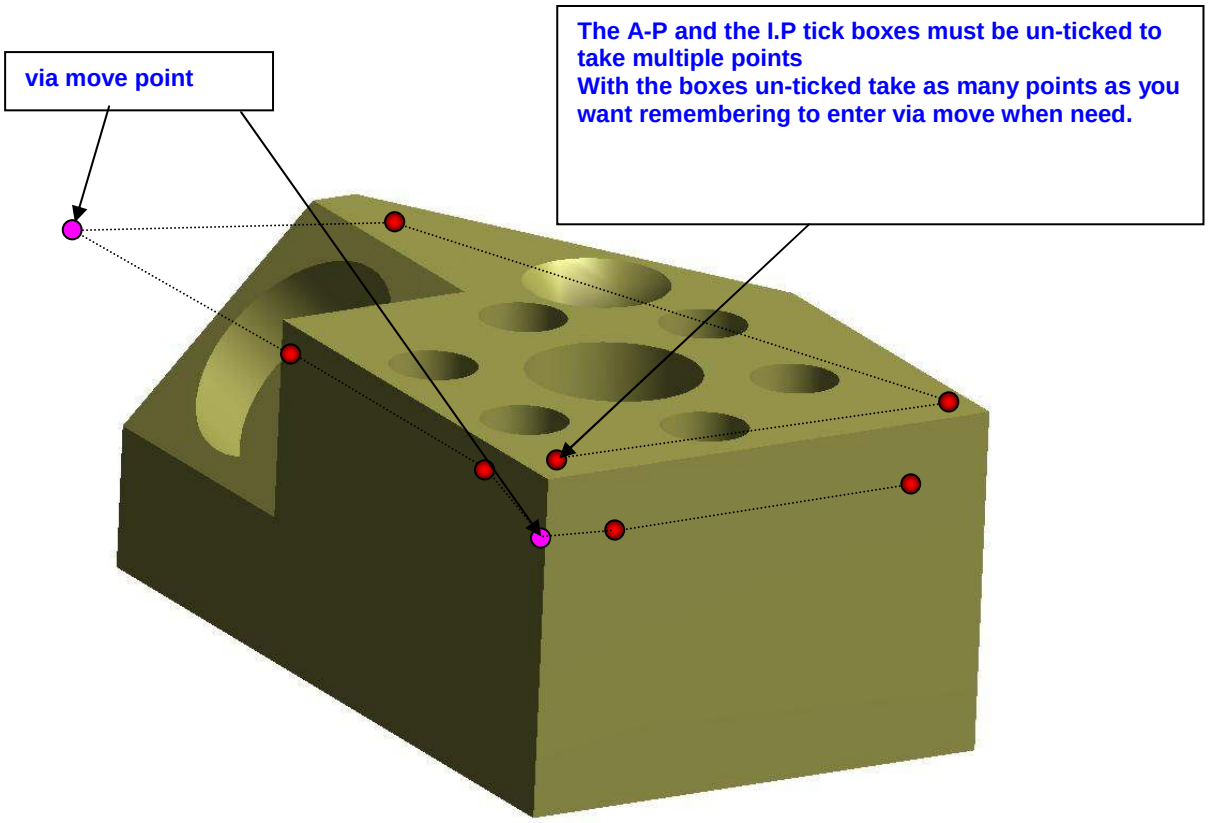


The A-P and the I.P tick boxes must **not be** ticked to take multiple points

As you measure points on the part they are listed below in the curve measurement window.

The machine will always take the shortest path between points so if you need to move around some thing you must enter via moves to stop the machine crashing.
You have to do this because the probe only moves to the safety zone between different measured features.

www.EngineeringBooksPdf.com



www.EngineeringBooksPdf.com

16.0 ERASING DATA POINT

If you have accidentally probed on the wrong place on the component, or are unhappy with a point, you can always erase either an individual point or all the stored points. The methods for erasing data points are common to all measurement types.

16.1 Retake

Fig. 16.1-1 The 'Retake' button

The 'Retake' button is used to erase a single measurement point. By default this will be the last taken point, however to erase a different individual point click on the down arrow situated just above the cancel button.

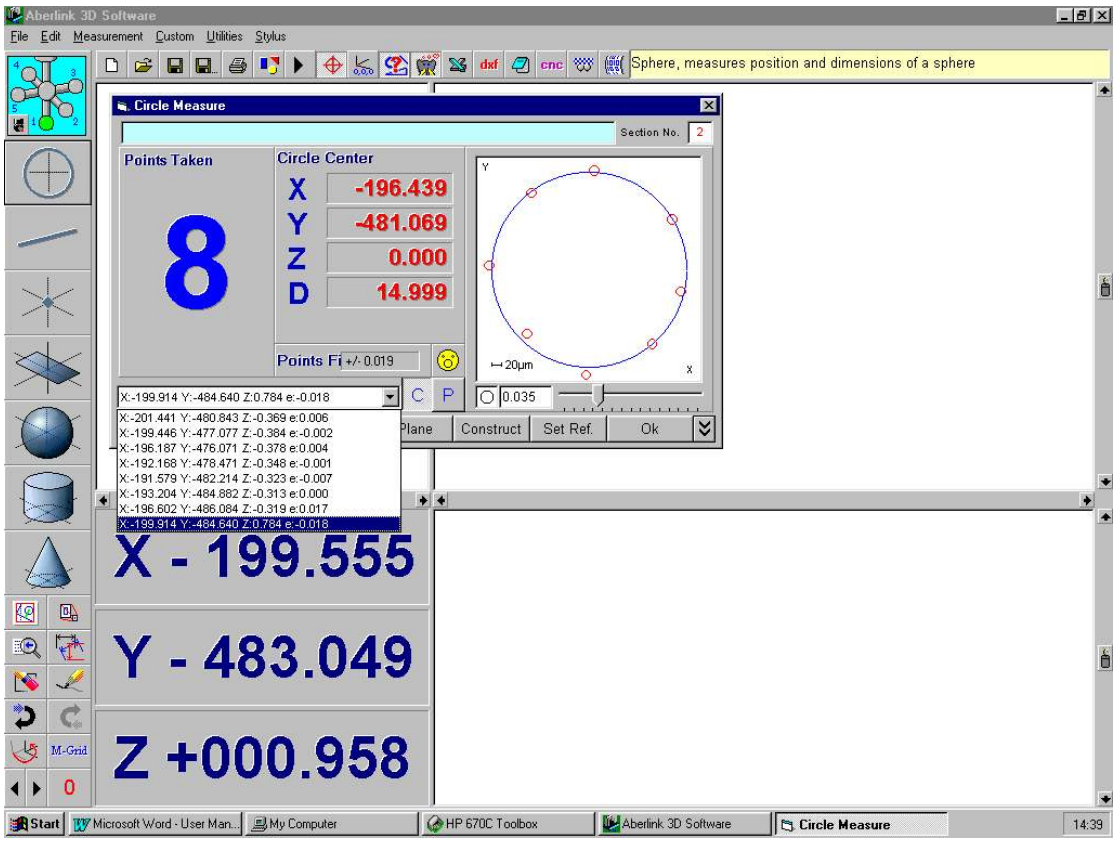


Fig. 16.1-2 Showing the pull down list of measurement points taken

A list of the X,Y,Z values for all previously taken points together with an 'e' value representing the error of that point from the calculated best fit shape will now be displayed on a pull down menu. Highlight the desired point by clicking on it, and then click on 'Retake'.

www.EngineeringBooksPdf.com

The screen will now recalculate the feature as if that measurement point had not been taken, ie. the number of points taken will reduce by one, and the erased point will no longer be shown on the graphical representation or taken into account in the displayed data.

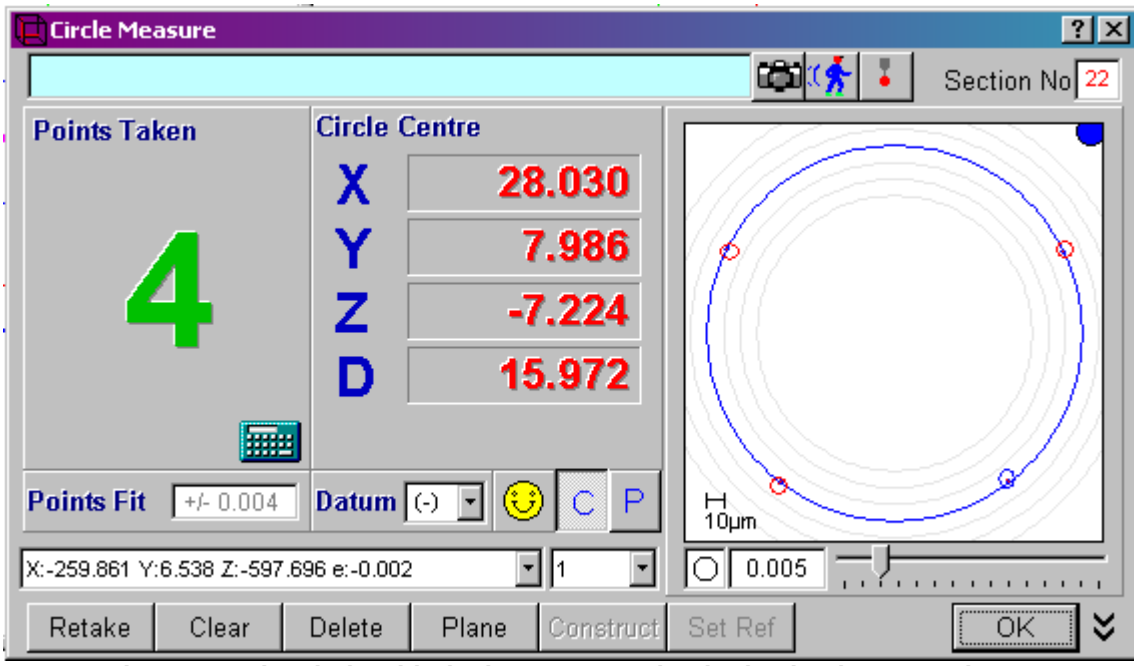


Fig. 16.1-3 The circle with the last measured point having been re-taken

16.2 Clear

Fig. 16.2-1 The 'Clear' button

To erase all the taken points on a particular measurement simply click on 'Clear'. The number of points taken will now return to zero, and the measurement window will appear as when first opened.

www.EngineeringBooksPdf.com

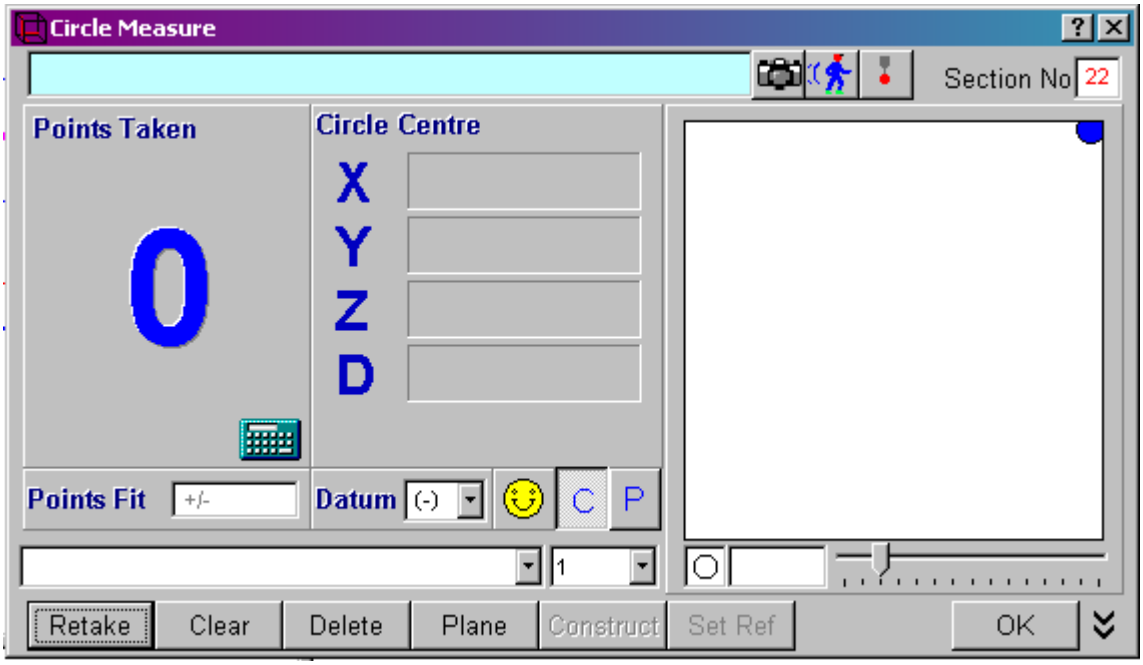


Fig. 16.2-2 A Circle Measure window having been cleared

16.3 Delete

Fig. 16.3-1 The 'Delete' button

Clicking on the delete button will erase all the taken points on a particular measurement and also close the measurement window and return to the main screen.

www.EngineeringBooksPdf.com

17.0 MANIPULATING THE GRAPHICS

During the inspection of a component, a pictorial representation of the part is built up on the screen in three 2-dimensional views. If a view is ever blank, it is probably because that view is focussed on the wrong position in the machine's measurement volume. In this situation you will need to zoom to the correct position.

This chapter explains how to manipulate the graphics windows in the main screen. These techniques can be used to adjust the size and position of the graphical representation on the screen at any time during an inspection.

17.1 Zoom Out

Fig. 17.1-1 The Zoom Out button

This button will make all 3 views zoom out by a fixed percentage.

17.2 Zoom Full

Fig. 17.1-2 The Zoom Full button

This button will make all three views zoom out to make the entire measurement volume fill the screen. The edge of the measurement volume will be represented by a red rectangle.

17.3 Zoom In

There is no button for zooming in. This is done by dragging the cursor, with the left mouse button depressed, to create a box around the desired area. When you take your finger off the mouse button the screen will zoom to the size of the box drawn.

www.EngineeringBooksPdf.com

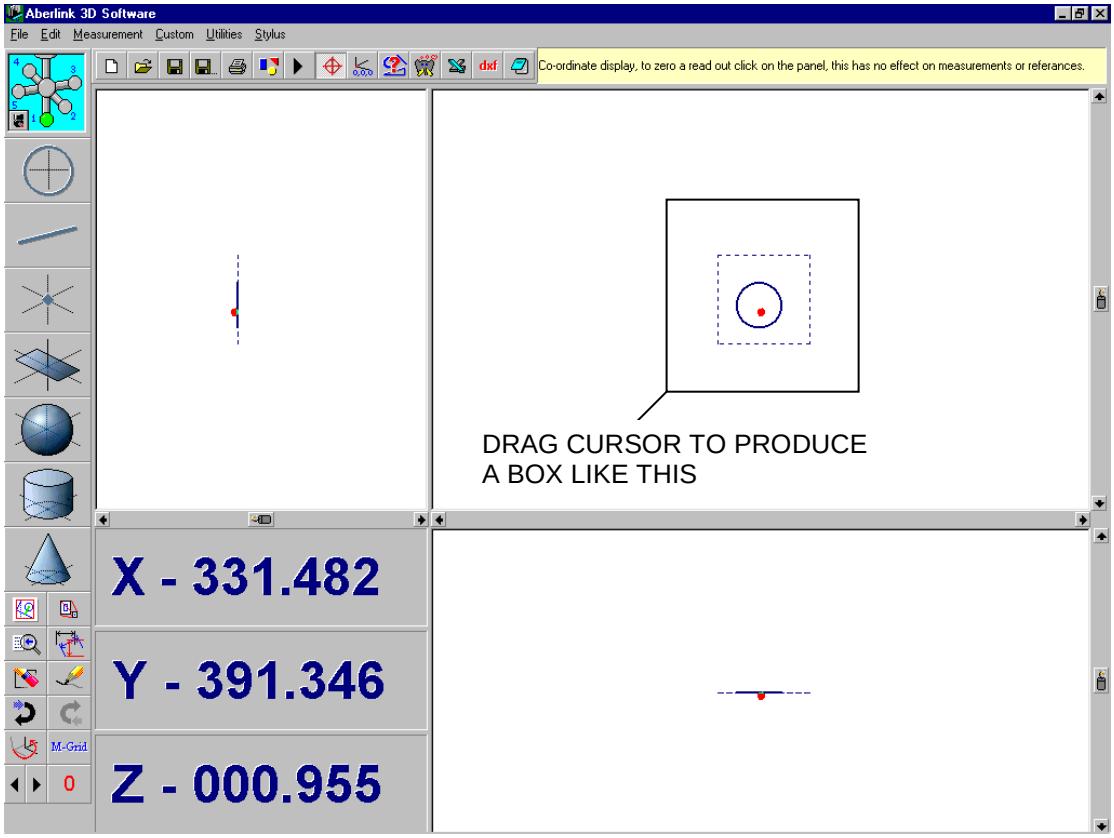
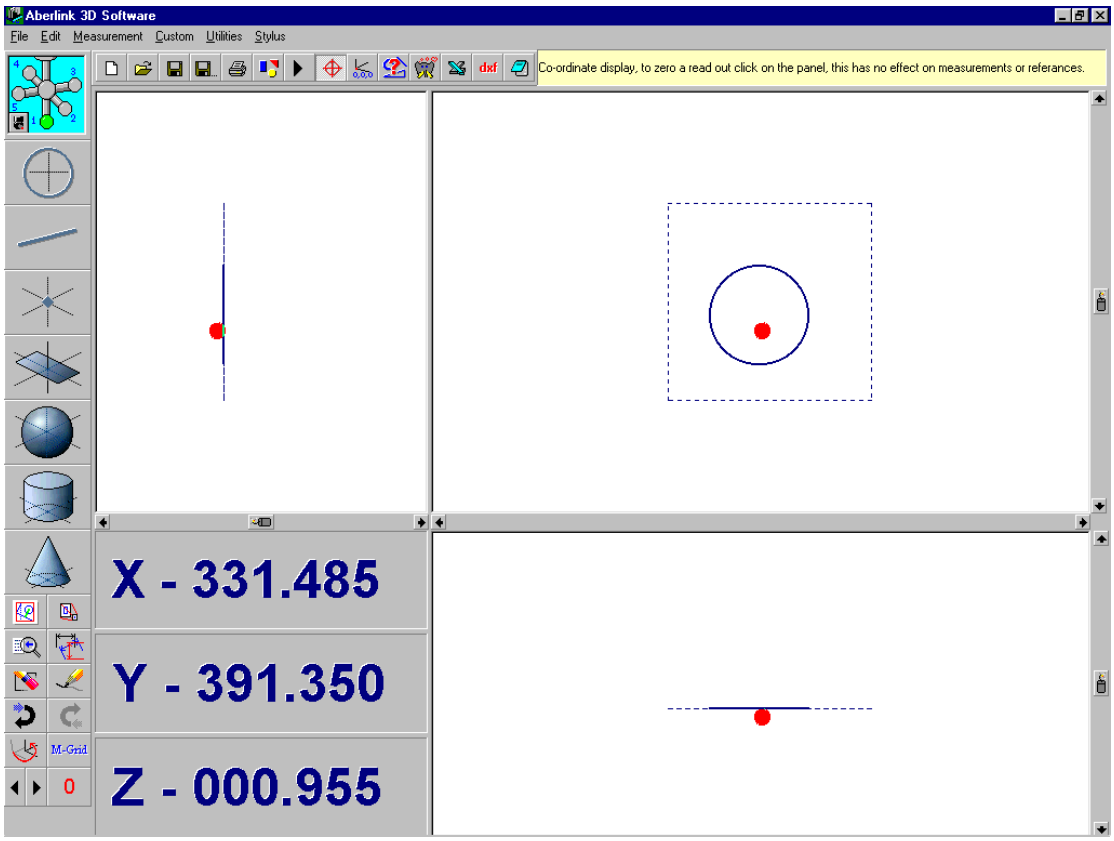


Fig. 17.1-3 The Main Screen showing a Zoom box created by dragging the mouse



www.EngineeringBooksPdf.com

Fig. 17.1-4 The Main Screen having zoomed in

17.4 Re-sizing & re-positioning the Graphics windows

It is possible to re-size & re-positioning the graphic windows by moving the mouse pointer over the place on the screen where the frames of the four graphic windows meet at the intersection between the windows the mouse point will change to a four headed arrow.
When you see the mouse pointer change you can click the left mouse button and drag the frames of the four graphic windows around the main screen changing the size of the individual graphic windows in the process.

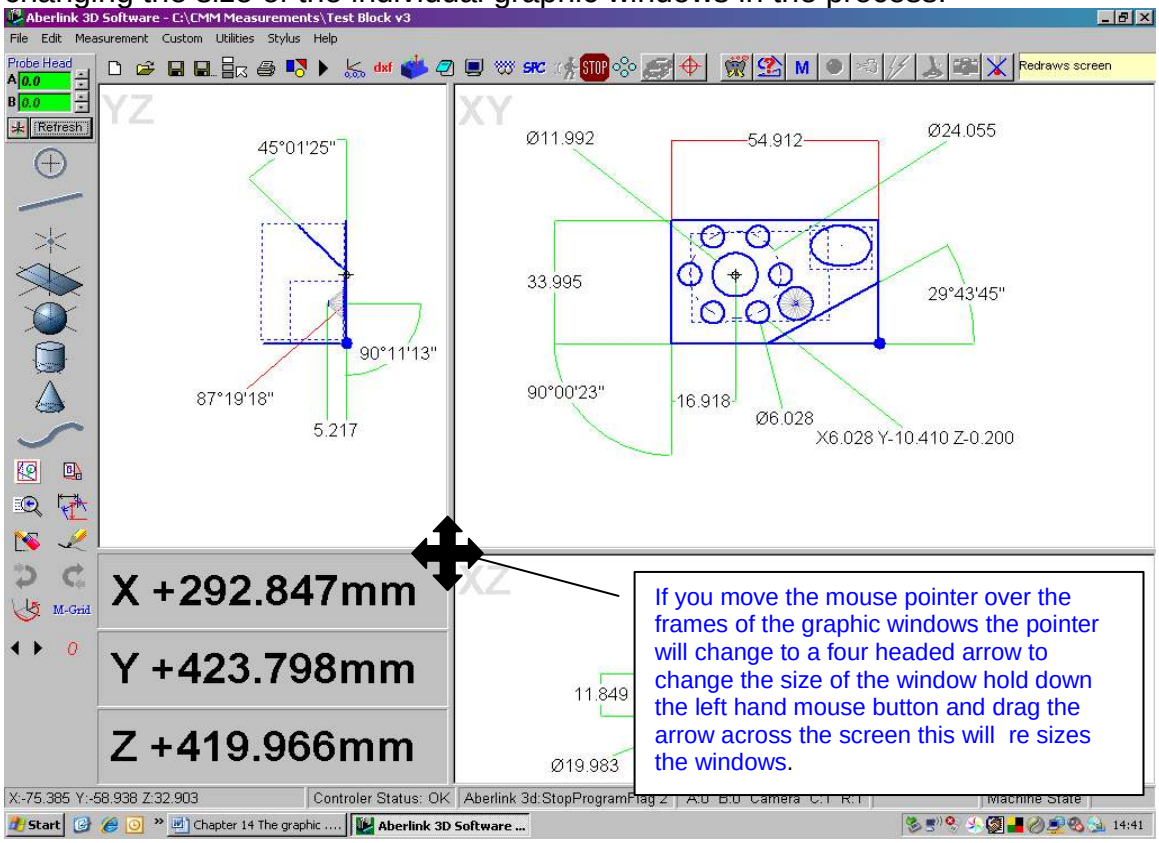


Fig. 17.4-1 Re-sizing the graphic windows

www.EngineeringBooksPdf.com

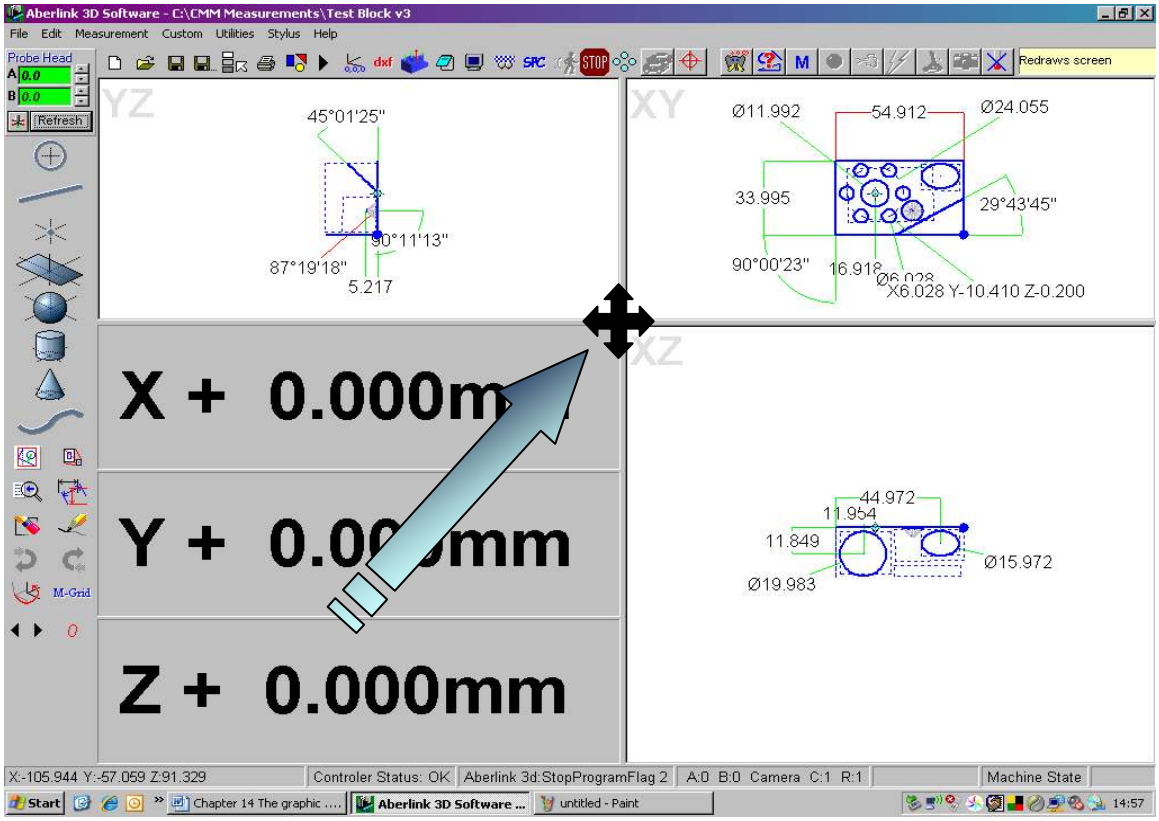


Fig. 17.4-1 Re-sizing the graphic windows dragging the mouse

17.5 Erase

Fig. 17.6-1 The Erase button

To erase a feature or a dimension from a graphics window, simply click on the ‘erase’ button to highlight it, and then click on the feature that you want to erase.

Beware though, because the ‘erase’ button is latching, and you need to switch it off when finished, by clicking on it again.

17.6Undo and Redo

Fig. 17.7-1 The Undo and Redo buttons

If at any stage you make a mistake, for instance you have erased more features than you intended, you can click on the ‘Undo’ button, which will undo the last function performed. In this case the last erased feature will reappear. Similarly if you have undone a function by mistake, you can click on

www.EngineeringBooksPdf.com

the ‘Redo’ button to bring it back.

17.7 Redraw

Fig. 17.8 The Auto Zoom Redraw button

This is probably the most useful zoom button of all. When you click on this button the screen will automatically zoom to display all the measured features in all 3 views, and also refresh the screen view.

www.EngineeringBooksPdf.com

18.0 SHIFTING, ROTATING AND CANCELLING REFERENCES

18.1 Applying Rotations or Shifts

It may be necessary during an inspection to apply a rotation or shift to the inspected features. This can be achieved by using the Grid Shift button:

Fig. 18.1-1 The Grid Shift button

Clicking on this button will bring up the following window:

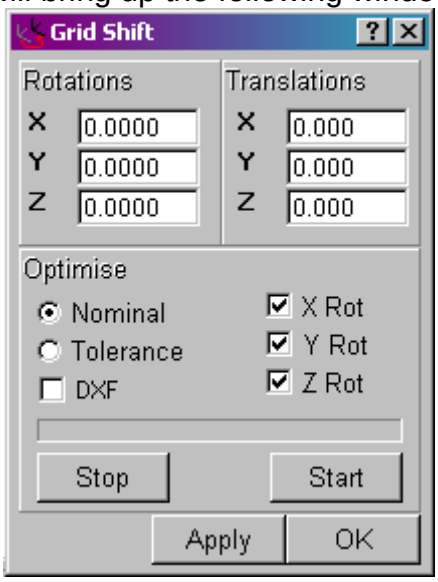


Fig. 18.1-2 The Grid Shift window

Now it is possible to apply a rotation or a translation to the measured features by typing the required angle(s) or distance(s) into the relevant box(es).

For instance, consider a component that consists of a plate with holes in its diagonal corners, which define a datum at 45 degrees. The best method for inspecting the plate is to measure the position of the 2 holes, construct a line between them and then set this line as a reference.

www.EngineeringBooksPdf.com

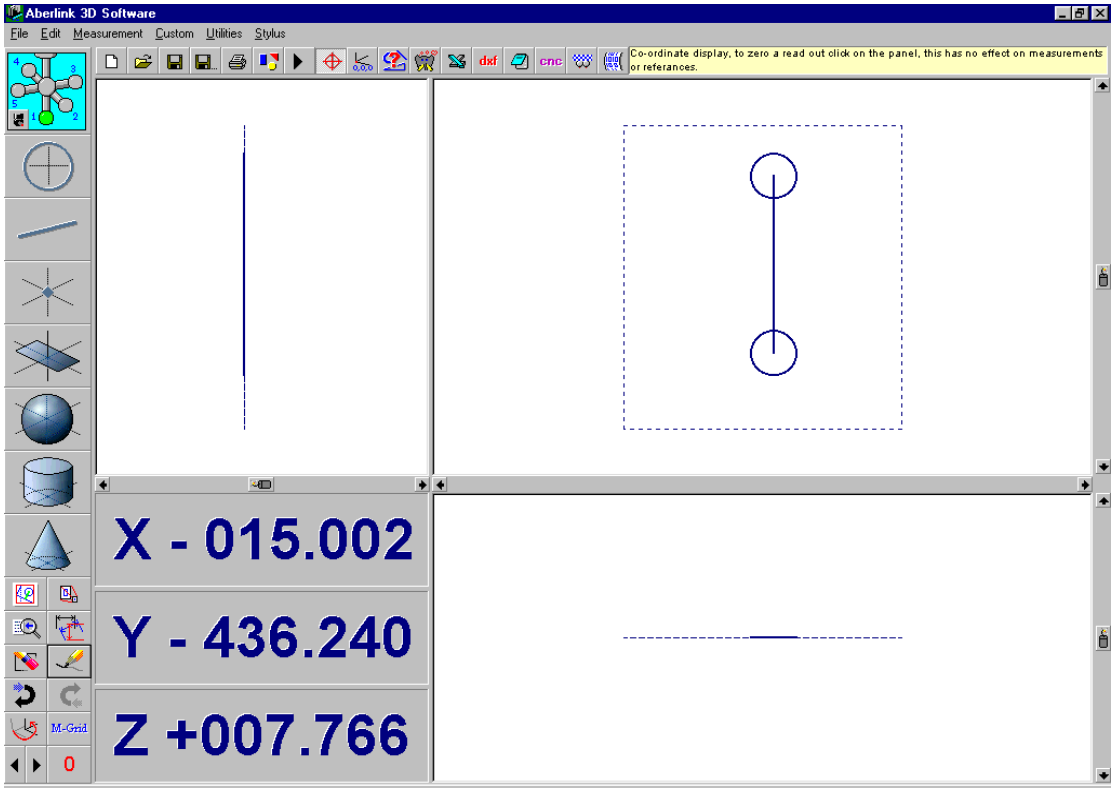


Fig. 18.1-3 The line between 2 holes having been referenced

Now click on the grid shift button, and type in 45 in the rotation about Z box. Click on 'Apply' followed by 'OK'.

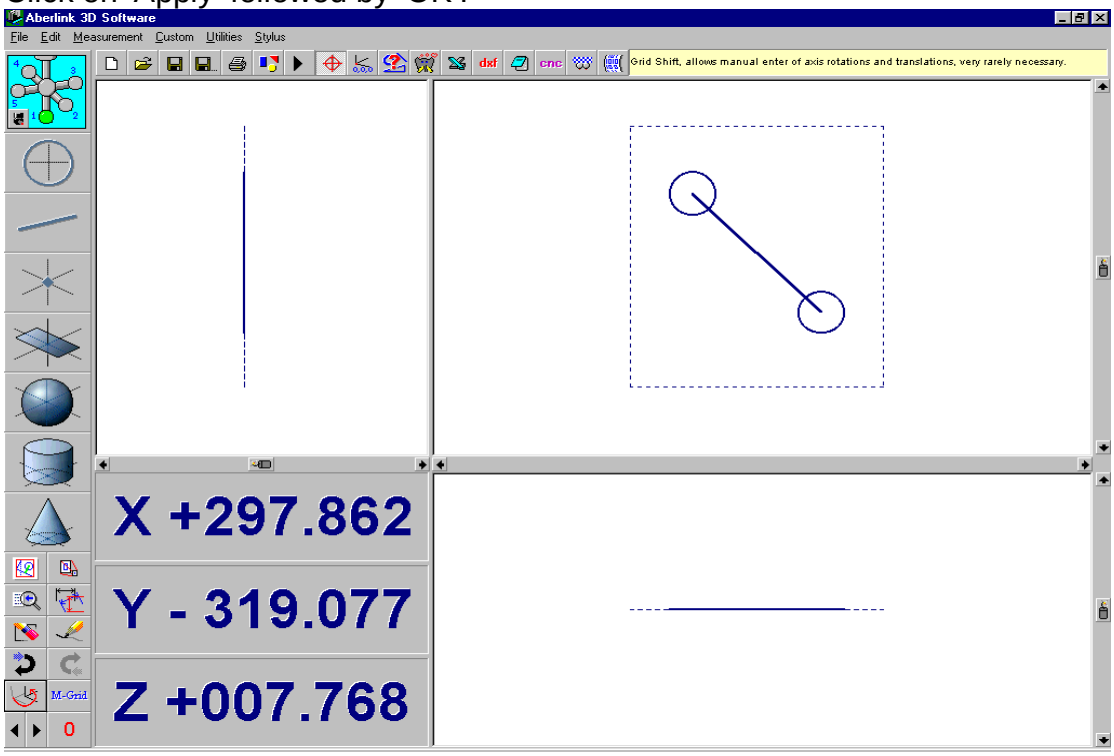


Fig. ***18.1-4 Two holes Aligned at 45 degrees***

The component will now be aligned at 45 degrees to the line between the 2 holes.

It is worth noting that when applying a rotation to inspection results, the

www.EngineeringBooksPdf.com

rotation will be performed about the origin set at the time. Also if rotations are required about more than one axis, then the order in which the rotations will be applied is Z first, Y next and then finally about X.

Applying translations to measured features works in exactly the same way. Note however that the rotations and translations are non-associative, ie. If you wish to apply a combination of translations and/or rotations to measured features, then they must all be applied at the same time.

18.2 Machine Grid (Cancelling References)

If at any point during an inspection you become confused with which references have been set, or about any rotations or shifts that have been applied, then click on the Machine Grid button

Fig. 16.2-1 The Machine Grid button

This will cancel all references that have been set, and all rotations and shifts that have been applied. The new datum position will be the machine's own reference position.

Note that this means that there will not be a reference plane set, and therefore any lines and circles that have been measured, will not be projected into a defined plane. In this situation any necessary references must now be reset.
Note:- You can not cancelling references after run through the program for the first time .

18.3 Optimising Alignment

It is possible with some complex shapes that the optimum alignment for the component is not achieved by referencing any one, or series of components. In this instance it is possible to select certain dimensions on the measured component, and then optimise the alignment of the component in order to minimise the errors of these dimensions.

Firstly select the dimensions that you wish to use to optimise the alignment of the component, by right mouse clicking on the dimension to bring up the dimension detail window.

www.EngineeringBooksPdf.com

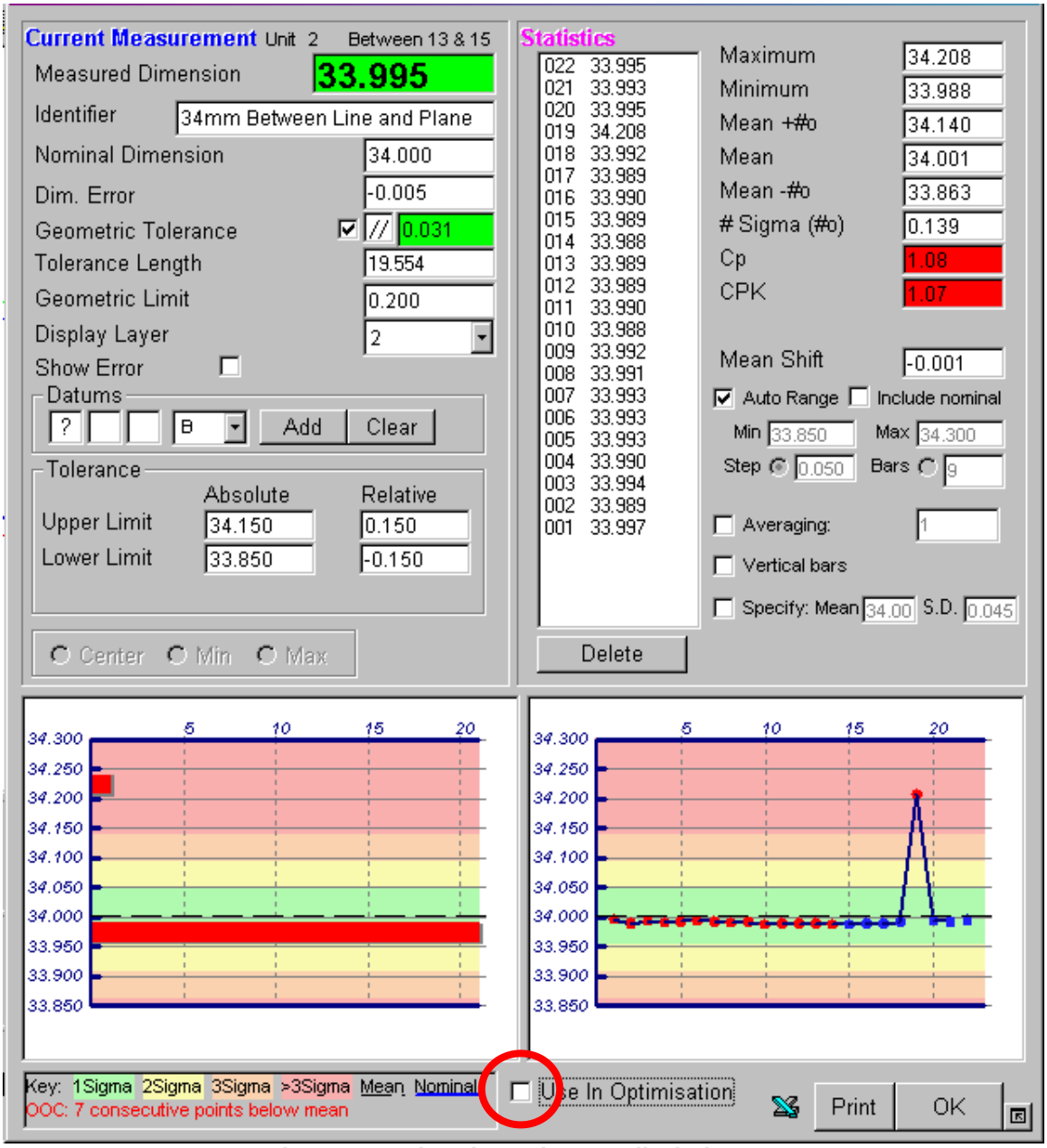


Fig. 15.3 –1 The Dimension Detail window

Change the nominal dimension to its correct value, if necessary, and then tick the box labelled ‘ Use in Optimization’. Click on OK to return to the main screen.

When you have selected all the dimensions that you wish to use in the optimisation, click on the Grid Shift button this will open the grid shift window, in the Optimise section of the grid shift window you have two modes of dimension optimization which you select by clicking on the small buttons next too Nominal / Tolerance.

18.3.1 Nominal optimization:-
If you select this option the software will translate & rotate the alignment system to get the optimum dimensions as close as possible to the nominal.

www.EngineeringBooksPdf.com

18.3.2 Tolerance optimization:-

If you select this option the software will translate & rotate the alignment system to get the optimum dimensions within the upper & lower tolerance for that dimension.

Note *If some of the dimension are more important than others you can bias the optimization by make their tolerance smaller, forcing any errors on to the other dimension.*

It is also possible to optimize the alignment with out rotating the part in X,Y or Z
To do this you need to switch off the X,Y,Z rotation options in the optimum dimensions alignment window by click the ticks on or off in the 'tick boxes' next to X Rot, Y Rot, Z Rot

After you have select the option you want click on the 'start button'
The software will take a few seconds to calculate the best alignment, and when it has finished click on OK.

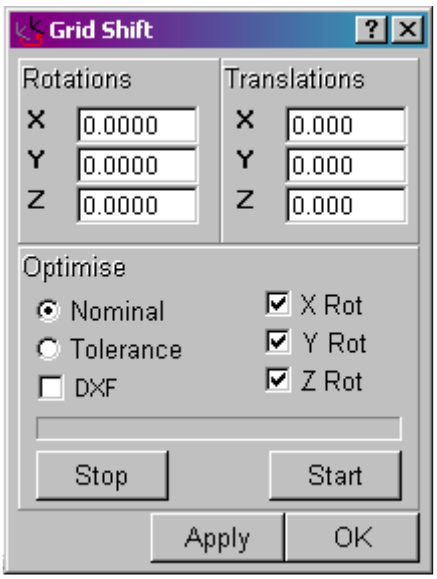


Fig. 18.3 –2 The Dimension Detail window

Note that the minimum number of dimensions should be selected that best define the rotation required for the component, ie. Typically this will be one dimension for X, one for Y and one for Z. This will help to minimise the calculation time required, although this is primarily dependant on the number of features that have been measured.

www.EngineeringBooksPdf.com

19.0 BRINGING UP DIMENSIONS ON THE SCREEN

One of the features of the Aberlink 3d software is that the dimensions for the measured component can be brought up on it's graphical representation, in the same way as they are shown on the component's engineering drawing.

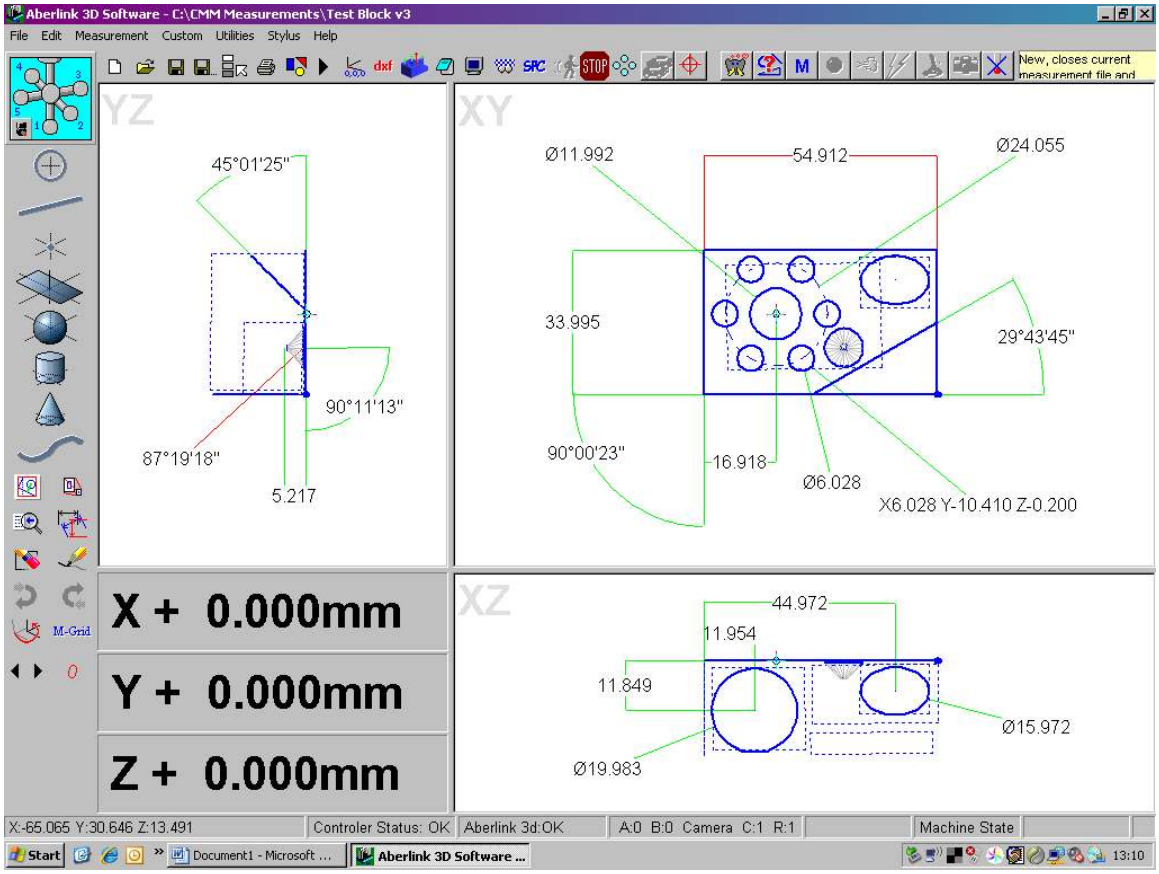


Fig. 19.0-1 Example of a dimensioned graphical representation

To bring up the dimensions onto the screen all you have to do is select the features that you wish to measure between, by clicking on their graphical representation. If the software doesn't default to the type of measurement that you want (horizontal, aligned etc.), then you will also have to select the correct dimension type.

In most cases the order in which you select the features will make no difference. The instances in which the order of selection does make a difference are highlighted later in this chapter.

It is important to grasp the following principles when bringing up dimensions:

1. Circles and spheres are treated as single points in space, with all dimensions being to their centre.
2. Cylinders are treated as 3 dimensional lines along their axis. When dimensioning between 2 lines (or cylinders etc.) the order of their selection is important. (see section 16.5 – Dimensioning between two Lines or Planes)

www.EngineeringBooksPdf.com

3. Cones may be treated either as a line along it's axis, or as a point at it's apex, depending on whether the cone is selected as the first or second feature. (see section 16.6 – Dimensioning to a Cone)

4. When linearly dimensioning a feature to a plane, the dimension will be perpendicular to the plane. When dimensioning between 2 planes the order of their selection is important. (see section 16.5 – Dimensioning between two Lines or Planes)

19.1 Selecting Features

When bringing up dimensions on the screen, it is necessary to select the features that you wish to dimension in the graphical representation.

To select a feature, click on the dark blue outline of the feature. Make sure that at the point that you click on the feature, there are no other overlapping features within the selection box.

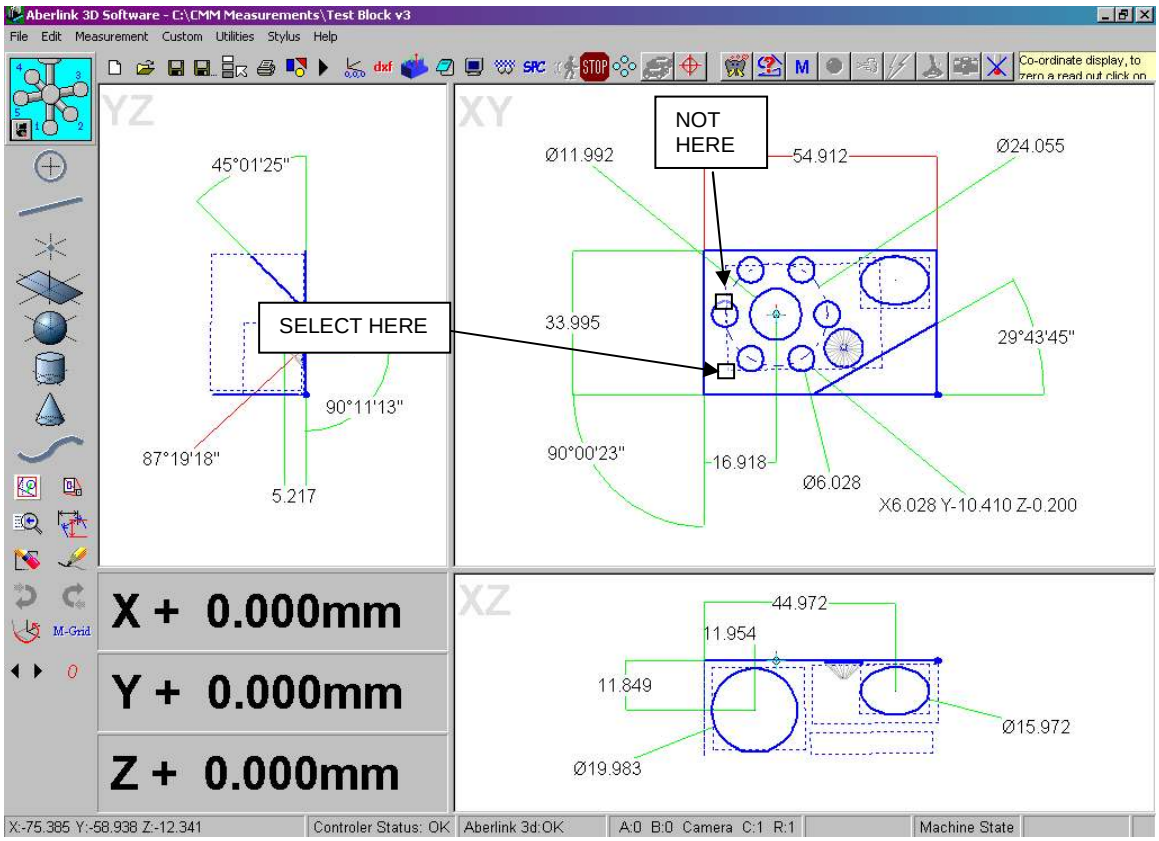


Fig. 19.1-1 Showing where to select a Feature

Note – You cannot select a 3 dimensional feature by clicking on the grey lines that represent its depth.

Also Note – If you are dimensioning in the side view of a reference plane, which has had 2 dimensional features projected into the plane, then a single line on the graphics will represent both the plane and the features. In this instance clicking on the line may select any one of the features. To define the

www.EngineeringBooksPdf.com

specific feature required you must use the feature select buttons. See section 16.7 of this chapter.

The feature selected will become pink.

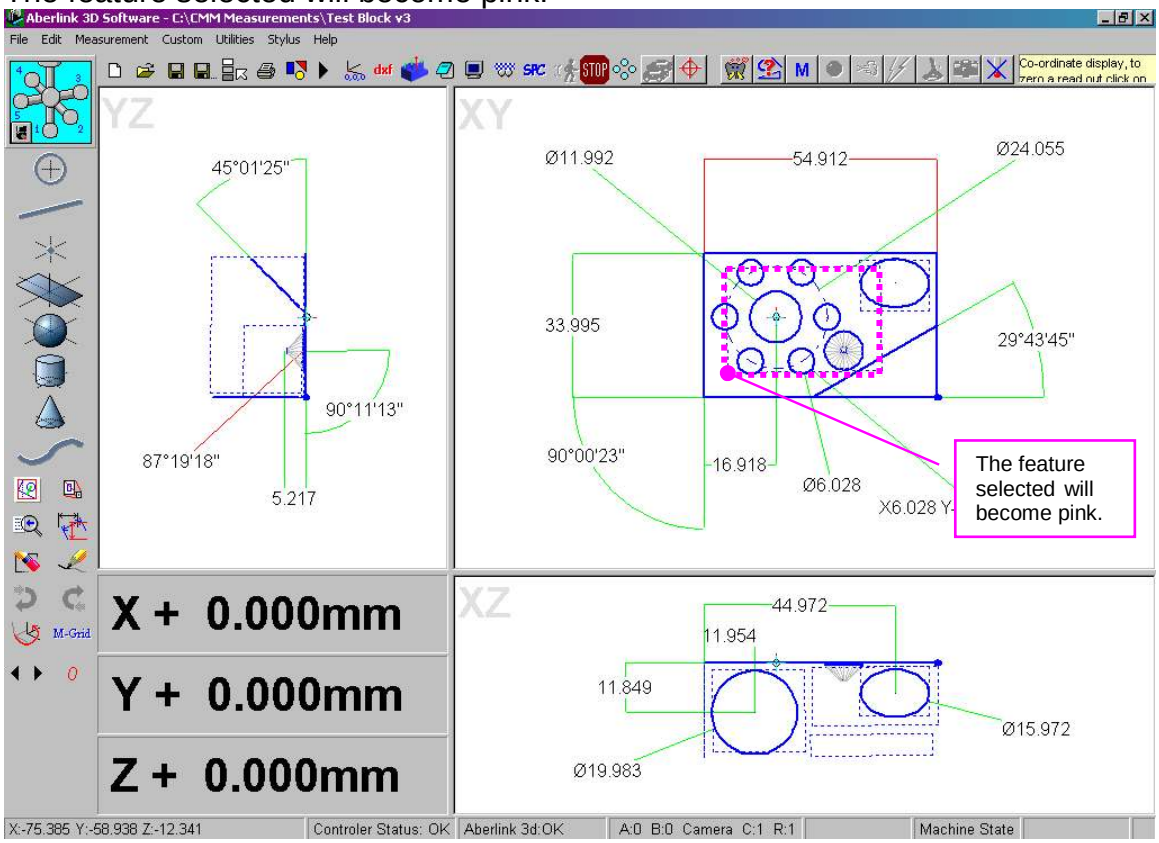


Fig. 19.1-2 Showing a Feature that has been selected

- If you have trouble selecting a feature there are various courses of action.
- You can use the feature select button to assist selection of the feature. See section 19.2 of this chapter.
- You can zoom in more closely onto the feature, to make it easier to select. See chapter 14 – Manipulating the Graphics.
- You can change the size of the selection box, in the software set up. See chapter 23 – Software set up.

19.2 Using the Feature Select Buttons

In some instances it will be difficult to select a feature by clicking on it directly. For instance, the same line will represent both the side view of a circle, and also the plane into which it is projected. If you have difficulty in selecting a feature, the Feature Select buttons will provide the solution.

Fig. 19.2-1 The Feature Select buttons

www.EngineeringBooksPdf.com

The red number (initially zero) refers to the feature number in the order that it was measured. For instance if you measured a plane first, followed by a circle and then a line, the plane would be feature number 1, the circle number 2 and the line number 3. Now you can use the right and left arrows to increment and decrement the feature number, to obtain the feature desired.

So that you don't have to memorise the order that the features are taken in, as the feature number is selected, the graphical representation of that feature will become red.

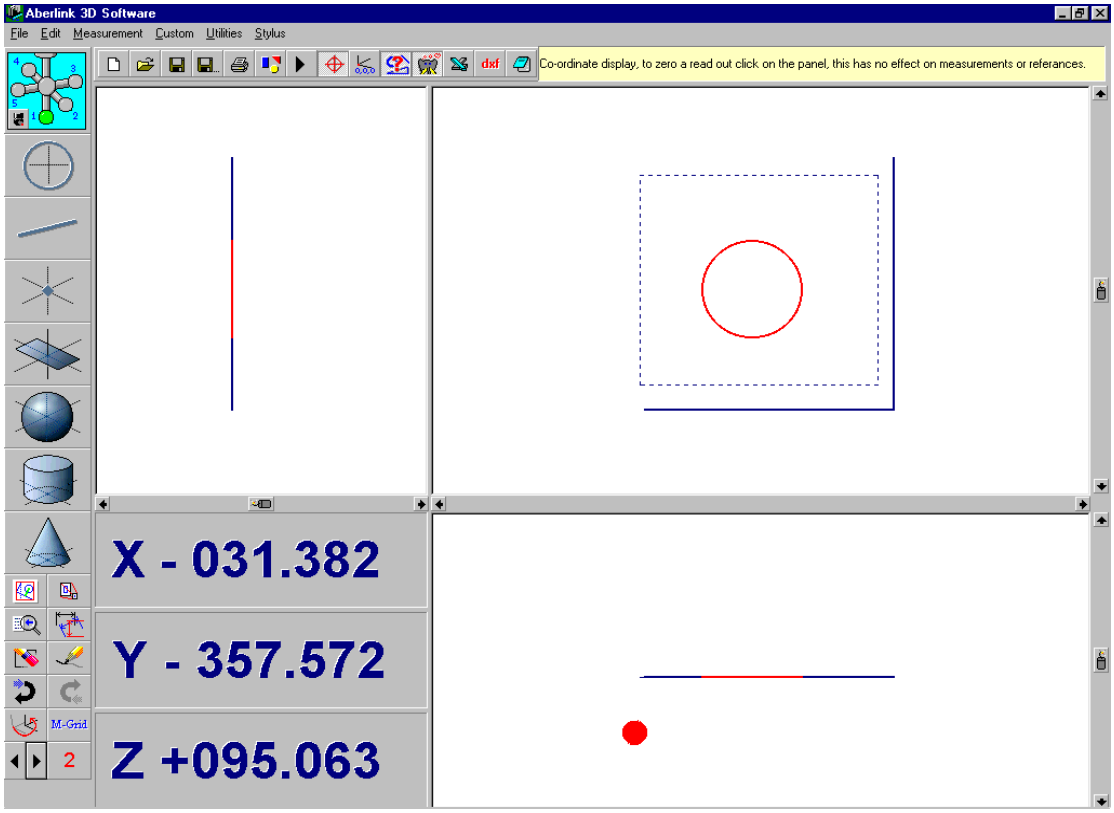


Fig. 19.2-2 Showing a circle that has been highlighted using the Feature Select buttons

Now click on any bit of blank background within the view that you are dimensioning, and the desired feature will be selected.

www.EngineeringBooksPdf.com

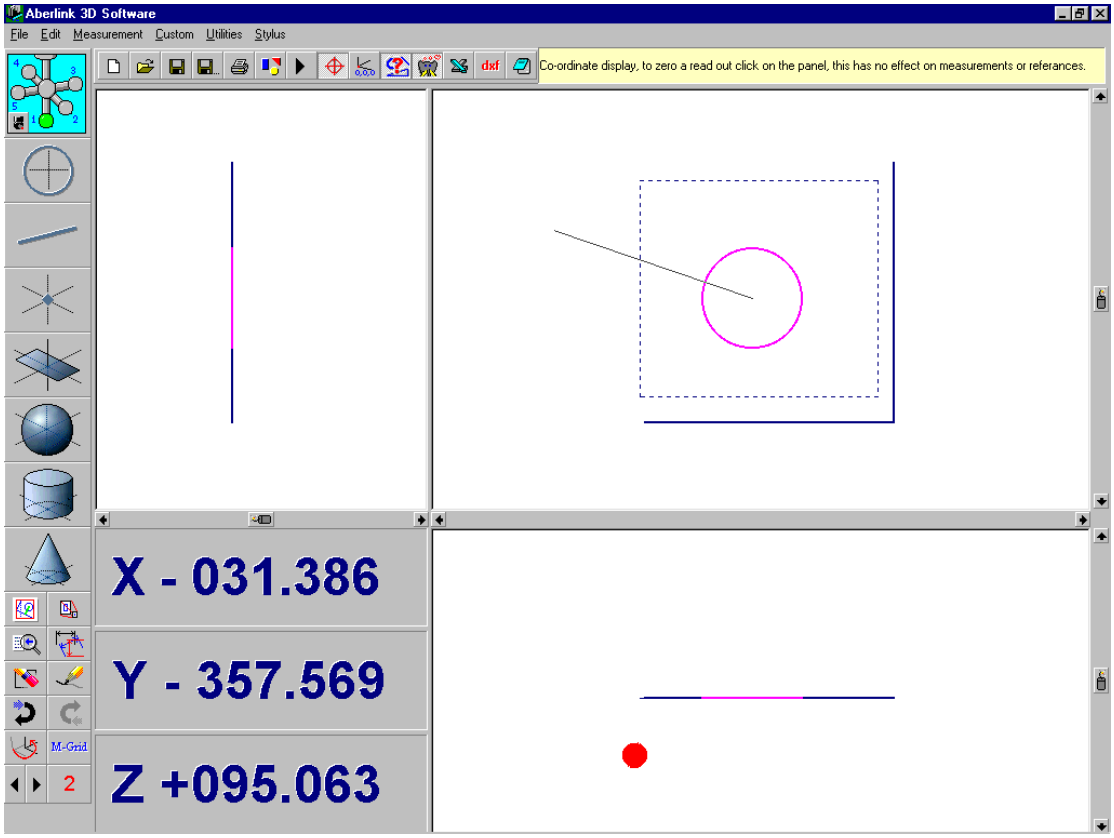


Fig. 19.2-3 Showing a circle that has been selected

Note – When using the Feature Select buttons you must click on blank background to ensure selection of the desired feature. Clicking on an alternative feature may still select it.

The Feature Select button will only work to select a feature once. If you need to select the same feature twice (for instance to display the diameter of a circle), simply click on the red number again to re-highlight that feature. (This removes the necessity to use the right and left arrows to move off, and return to the desired feature). Now click on blank background again to re-select the feature.

19.3 Aligned Dimensions

19.3.1 Three Dimensional

An aligned dimension can be described as either the shortest distance between two points in space, or as the perpendicular distance between a line and a point. If the 2 features being dimensioned are not in the same plane then this will produce a 3 dimensional aligned dimension.

When selecting two points (or circles or spheres) to dimension between, or a line and a point (or equivalent) the software will automatically default to producing a 3 dimensional aligned dimension.

www.EngineeringBooksPdf.com

Now select the second feature to which you wish to dimension. Again, if correctly selected the second feature will also turn pink, and the outline of some aligned dimension leader lines will appear.

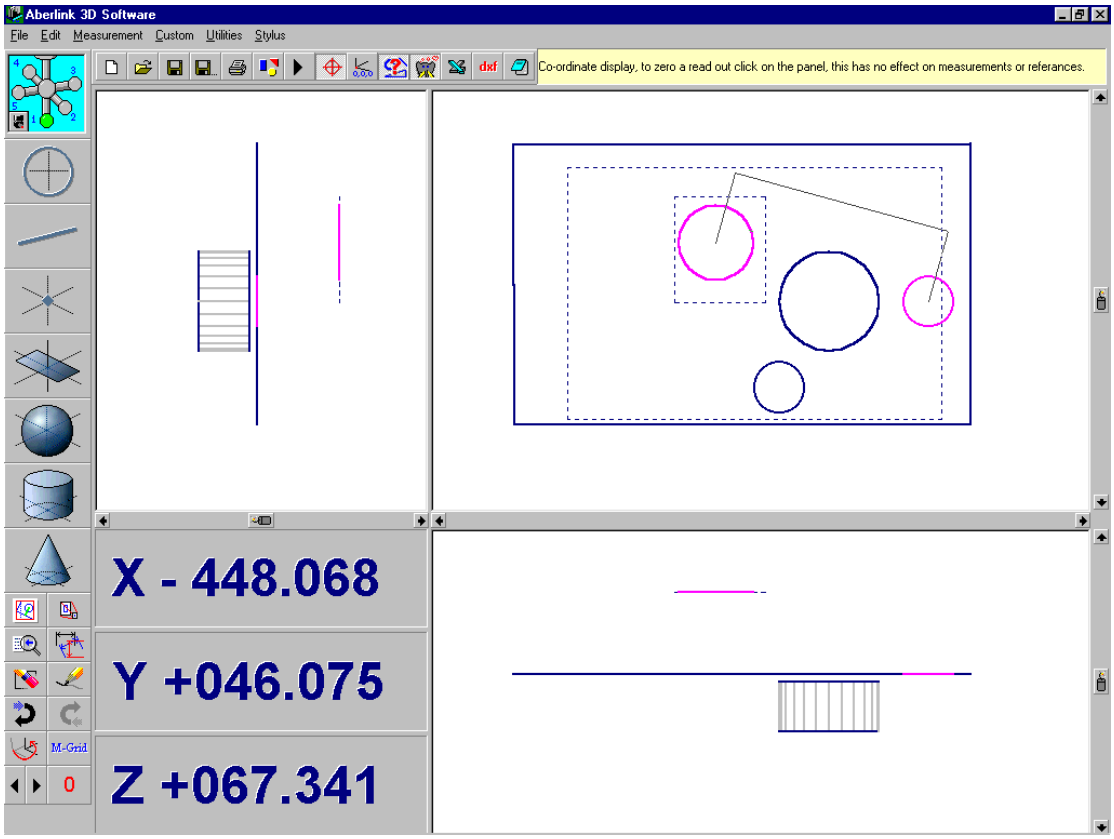


Fig. 19.3-1 Showing 2 Features that have been selected

These leader lines can be positioned at any length on either side of the features by moving the cursor on the screen. When they are at a convenient position click for a third time. The aligned dimension will now appear.

www.EngineeringBooksPdf.com

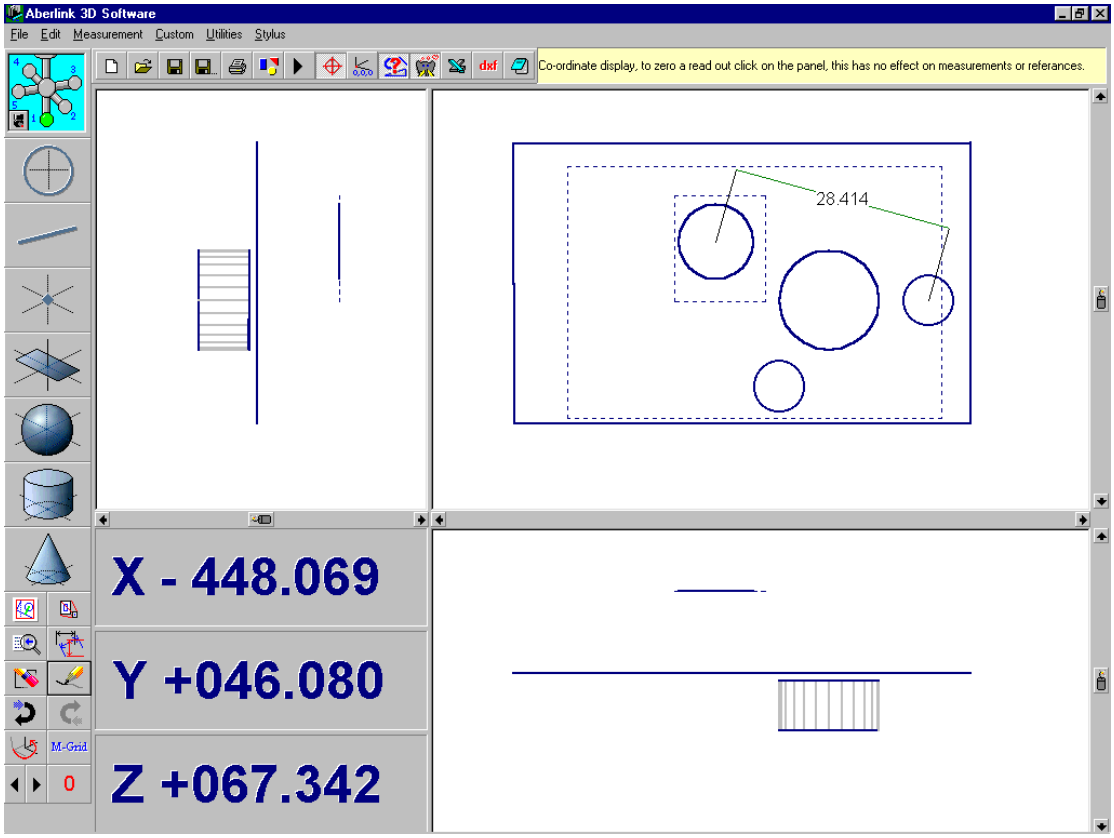


Fig. 19.3-2 Showing a 3D Aligned dimension between 2 circles

Note – If when trying to select the second feature you click on blank screen by mistake, (ie. the second feature is not selected correctly) the measurement will be aborted, and the first feature that was selected will return to it's normal dark blue colour. This can be a useful technique for aborting a measurement where the first feature has been selected in error.

Note that as the 2 circles being dimensioned are in different planes, the aligned dimension is 3 dimensional. This is best shown by bring up the same dimension in the other 2 views.

www.EngineeringBooksPdf.com

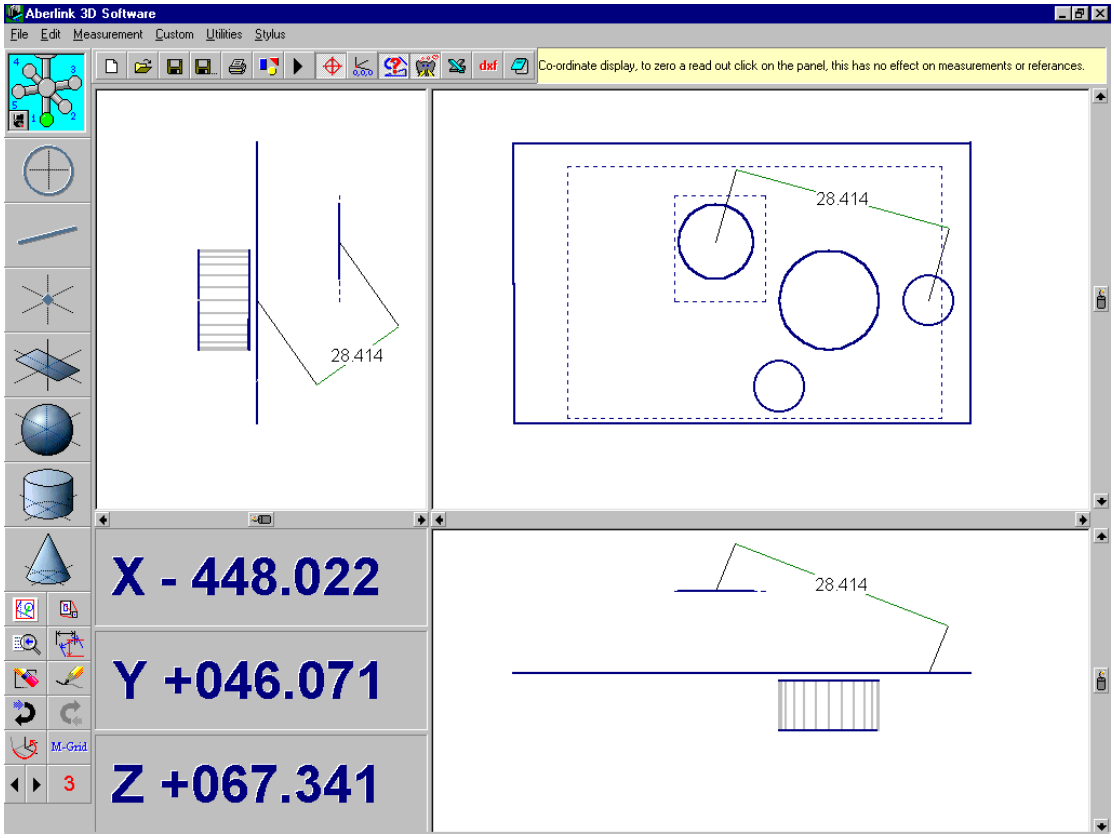


Fig. 19.3-3 Showing a 3D Aligned Dimension in all 3 views

19.3.2 Two Dimensional

It is possible to force aligned dimensions to be aligned only in the 2 dimensions of the view in which it is called up. For instance perhaps you are interested only in the X,Y separation of the 2 holes dimensioned above.

In this case select the 2 circle features as before, but this time before placing the dimension, click on the Dimension Type button on the side toolbar.

Fig. 16.3-4 The Dimension Type Button

This will bring up the following window:

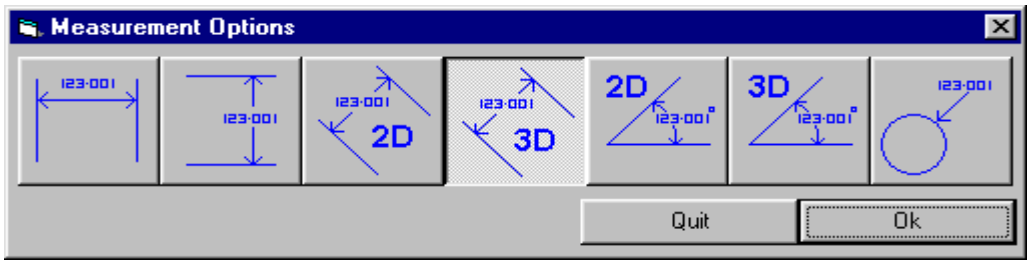
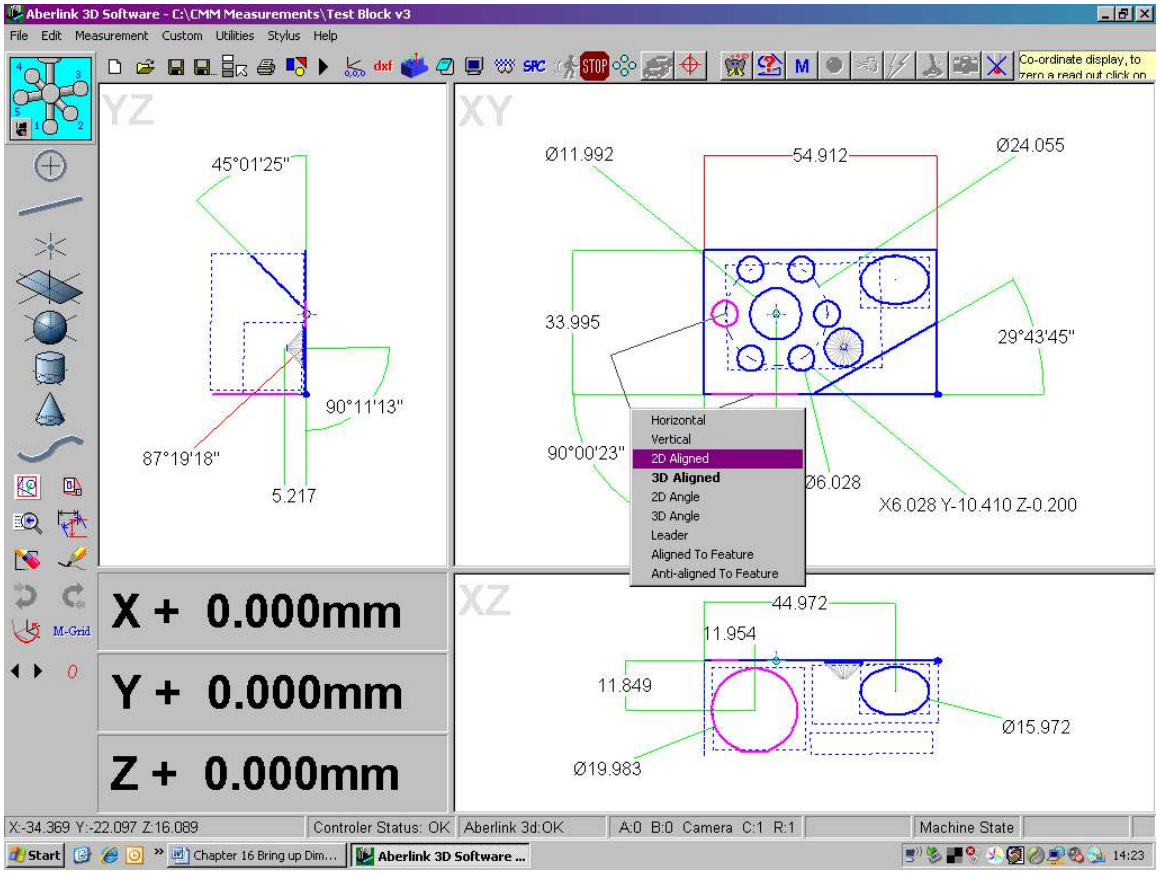


Fig. 16.3-5 The Dimension Type window

Click on the 2D aligned dimension button, and then click OK. The window will disappear, and you can now position the dimension as before.

www.EngineeringBooksPdf.com

An alternative method to one above is to use the ‘dimension type drop down menu’ to use this method you need to press the right hand mouse button after you have selected the 2nd feature, the drop down menu will appear giving a list of dimension types, select the 2D Aligned option



Then left hand click on the screen to place the dimension

Note - *Using the ‘dimension type drop down menu’ is the preferred method of change the dimensional type.*

www.EngineeringBooksPdf.com

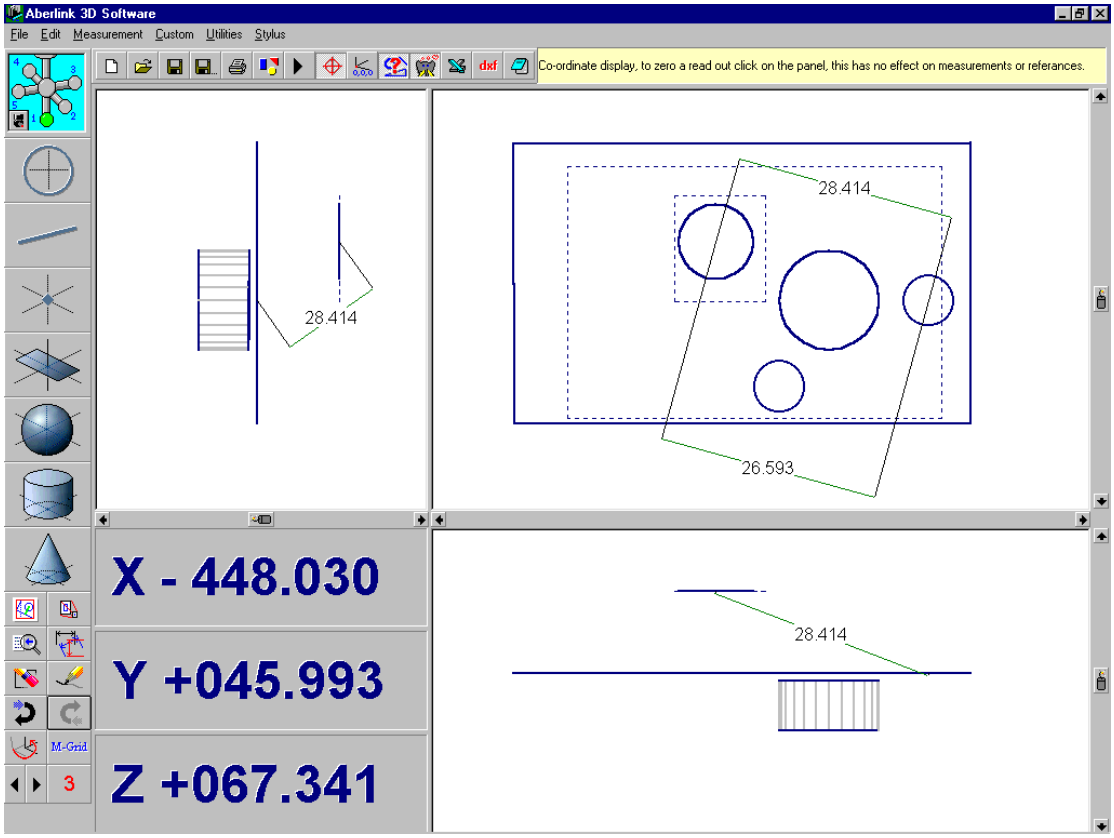


Fig. 19.3-6 Showing a 2D and a 3D Aligned Dimension between the same features

Also note -that the 2D and 3D aligned dimensions will appear identically in the 2 dimensional views, although having different values. Care should therefore be taken when dimensioning between features that are not in the same plane.

19.4 Horizontal and Vertical Dimensions

A horizontal or vertical dimension can be described either between two points in space, or between a line or plane and a point, in which case the dimension will be attached to the mid-point of the line or plane. Instead of being aligned between the features, these measurements will be perpendicular to the alignment of the component (which is defined by setting references, see chapter 5 – Aligning the axes of a component). If no reference has been set, then the axes of the machine will be used.

***Note** – If the alignment of the component has not been defined, and the horizontal and vertical dimensions are perpendicular to the axes of the machine, these measurements will have little or no significance.*

A horizontal or vertical dimension can also be described between two lines, or two planes, but in this case the dimension will attach to the mid point of the first line or plane selected only, and give their separation in the direction perpendicular to this line or plane. Dimensioning between two lines or planes is described in sections 19.5

www.EngineeringBooksPdf.com

When selecting two points (or circles or spheres) to dimension between, or a line and a point (or equivalent) the software will automatically default to producing an aligned dimension. A horizontal or vertical dimension must be defined as follows:

Select the features between which you wish to dimension, as for aligned dimensions in section 16.1 above. The features will turn pink and the outline of some aligned dimension leader lines will appear. Now before positioning the dimension, click on the Dimension Type button, which will bring up the Dimension Type window. The aligned dimension type will be highlighted by default, but click on the horizontal or vertical type button as required, and this button will now become highlighted. Now click on 'OK' and the software will return to the main screen with the outline of the dimension leader lines now showing the horizontal or vertical dimension, as selected.

These leader lines can be positioned at any length on either side of the features, as for aligned dimensions, by moving the cursor on the screen. When they are at a convenient position click the mouse for a third time and the dimension will now appear on the screen.

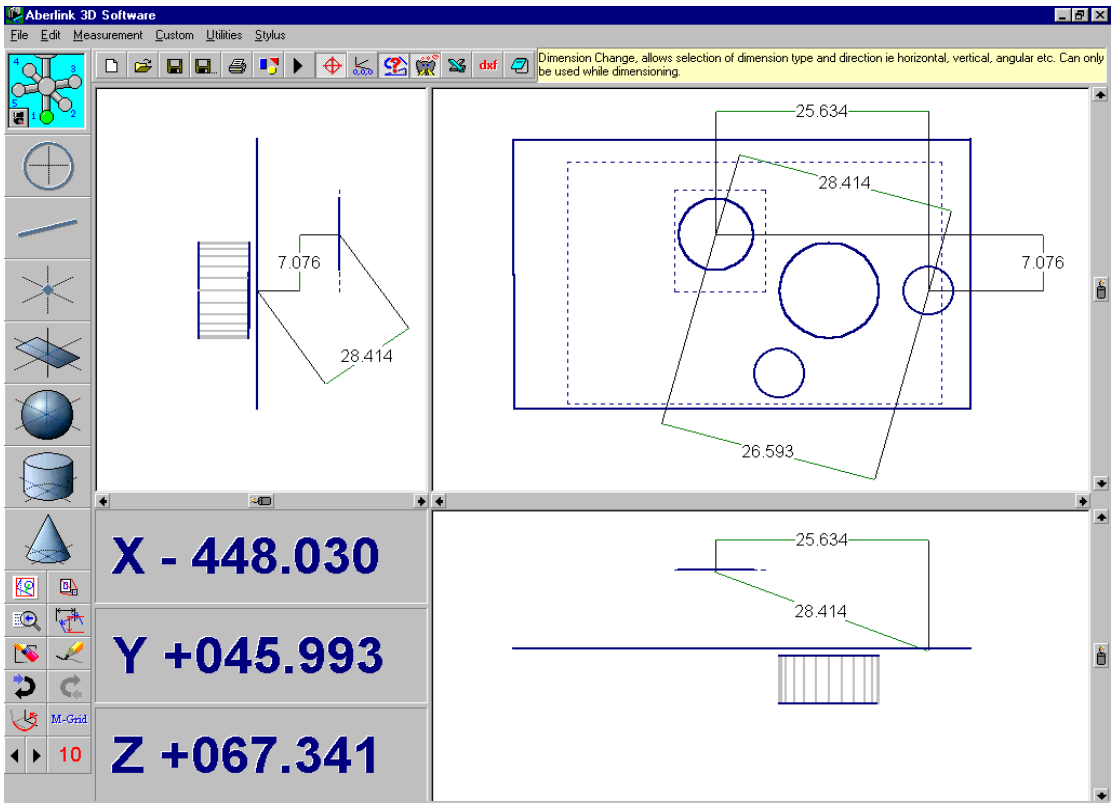


Fig. 19.4-1 Showing Horizontal, 2D Aligned and 3D Aligned Dimensions

19.5 Angular Dimensions

19.5.1 Three Dimensional

Angular dimensions may be created between two lines (or equivalent), or two planes, or between a line and a plane. Any combination of these features will

www.EngineeringBooksPdf.com

create an included angle, which may be dimensioned irrespective of the alignment of the component. As with aligned dimensions, if the features are not in the same plane, the angle created will be 3 dimensional.

Select the features between which you wish to dimension, as for aligned dimensions in section 16.1 above. The features will again turn pink if correctly selected. The software may now default either to an angular dimension or an aligned dimension, depending on what the angle between the features is, and the outline of the appropriate dimension leader lines will appear on the screen. If the software has correctly selected an angle dimension, the leader lines can be positioned at any length in any quadrant of the angle, by moving the cursor on the screen. When they are at a convenient position click for a third time and the dimension will now appear on the screen.

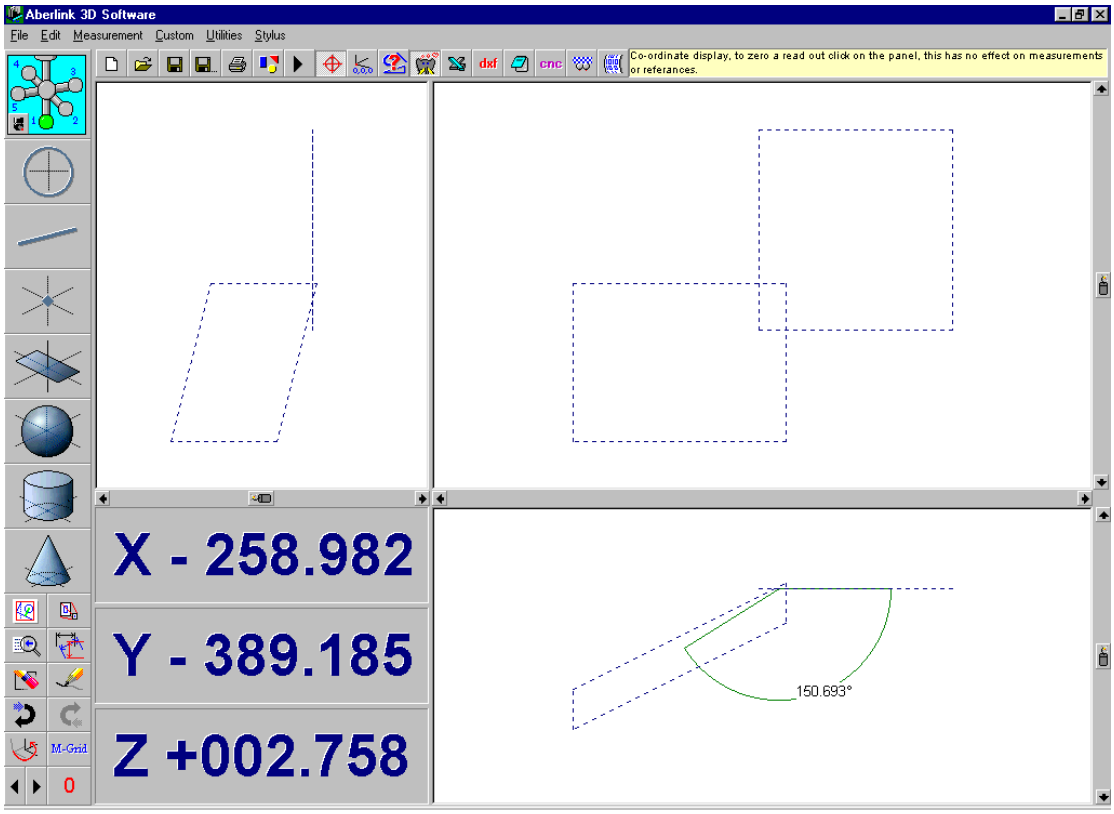


Fig. 19.5-1 A 3D Angular dimension between 2 Planes

If however, the software selected an aligned linear dimension, you must click on the Dimension Type button to bring up the Dimension Type window, prior to positioning the dimension. Click on the 3D Angular Dimension type button, and it will now become highlighted. Now click on ‘OK’ and the software will return to the main screen with the outline of the angular dimension leader lines showing. You can now position the dimension as previously described.

19.5.2 Two Dimensional

www.EngineeringBooksPdf.com

It is possible to split angles into the 2 dimensional components of the view in which it is called up. For instance perhaps you are interested only in the X,Y component of the angle between the 2 planes dimensioned above.

Once again use the Dimension Type button to select a 2D angle dimension.

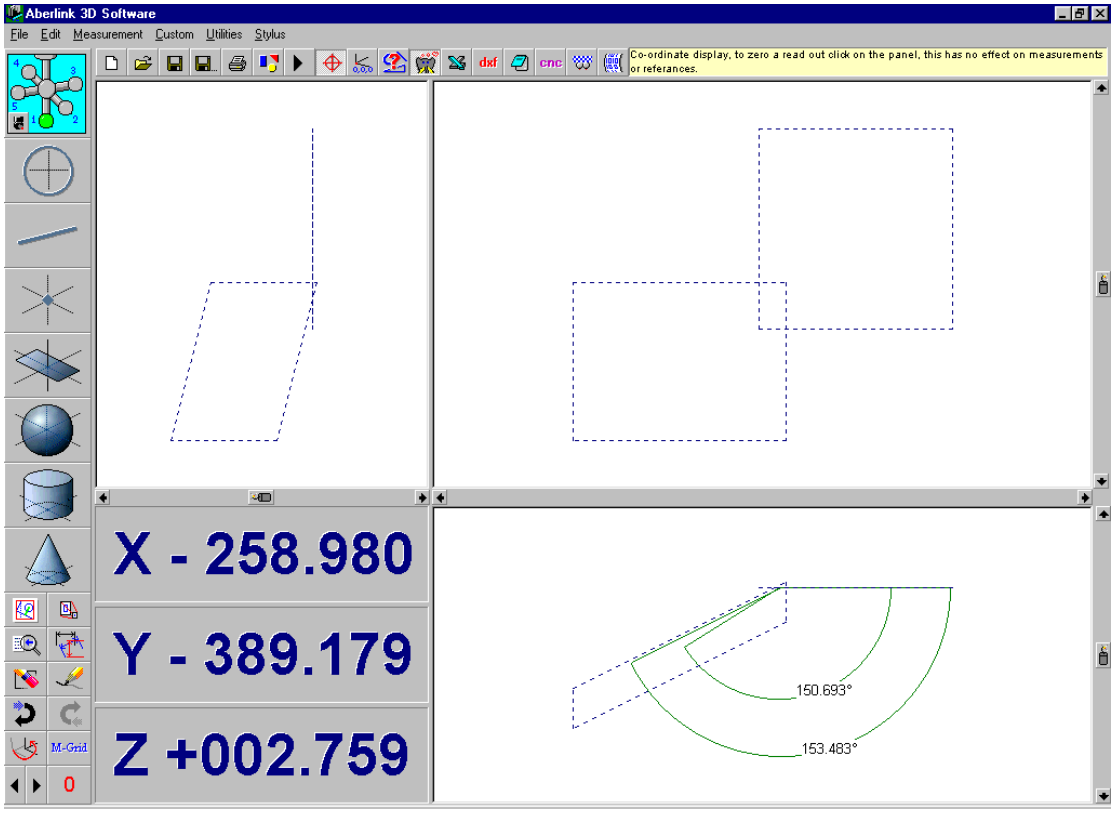


Fig. 19.5-2 2D and 3D Angular Dimensions between 2 Planes

Again note that the 2D and 3D angle dimensions will appear identically in the 2 dimensional views, although having different values. Care should therefore be taken when dimensioning between features that are not in the same plane.

19.6 Diameter Dimensions

The diameter of a circle or a cylinder may be brought up on the screen as follows:

Select the circle, sphere or cylinder as previously described. If correctly selected the feature will turn pink.

Note – You cannot select a cylinder by clicking on the grey lines that indicate its depth.

Now select the same feature again. The outline of a single dimension leader line will appear on the screen, and this can be positioned anywhere on the

www.EngineeringBooksPdf.com

screen by moving the cursor. When it is at a convenient position, left click the mouse for a third time and the diameter will now appear on the screen.

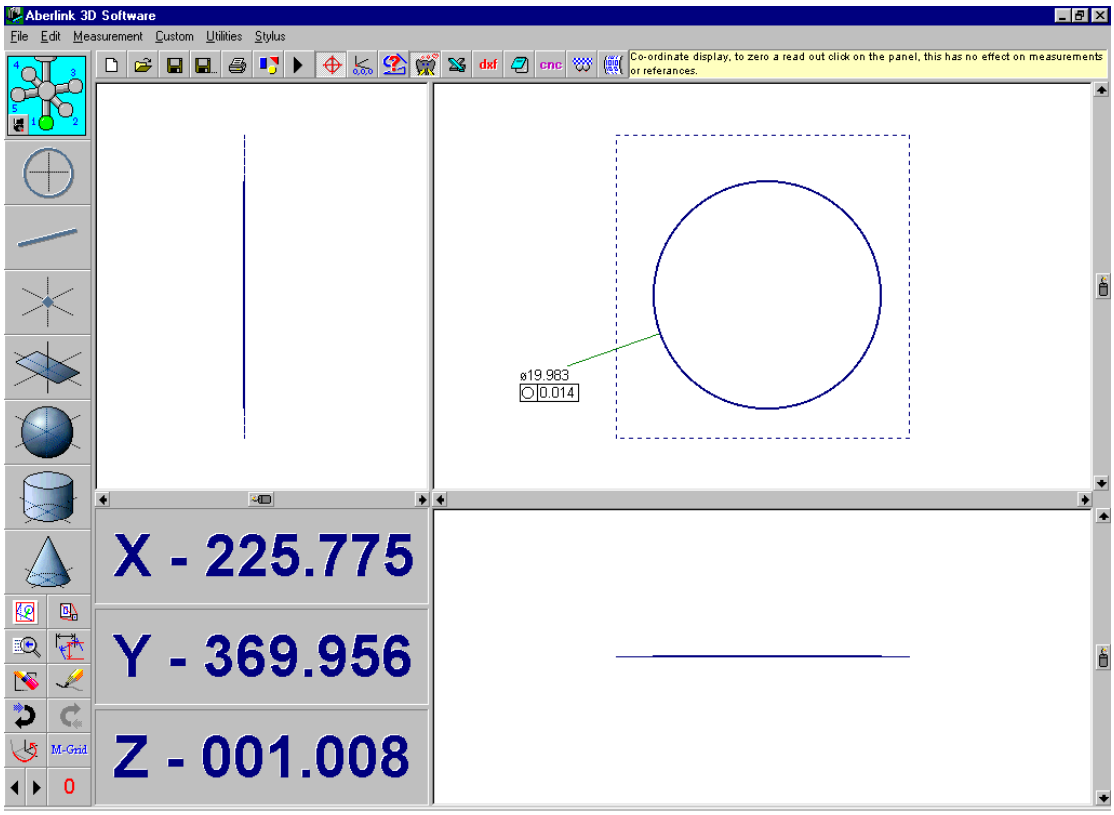


Fig. 19.6-1 A diameter dimension

19.7 Dimensioning between two Lines or Planes

A linear dimension may be produced between two lines or planes by attaching the dimension to the first line or plane selected, and extending it either perpendicularly (for an aligned dimension), or horizontally or vertically (for horizontal and vertical dimensions), until it reaches the mid-point of the second one.

If the lines or planes are not parallel, different dimensions will be achieved depending on which line or plane is selected first. Also the value for run out is calculated by projecting perpendicular construction lines from the first selected line, to meet the ends of the second line. The run out (or perpendicularity) will be the difference between these two values.

Selecting the long line first:

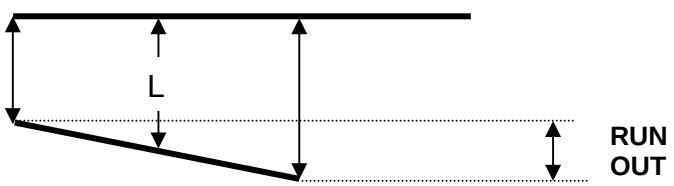


Fig. 19.7-1 Showing the separation and run out between the long line and the short line

www.EngineeringBooksPdf.com

Selecting the short line first:

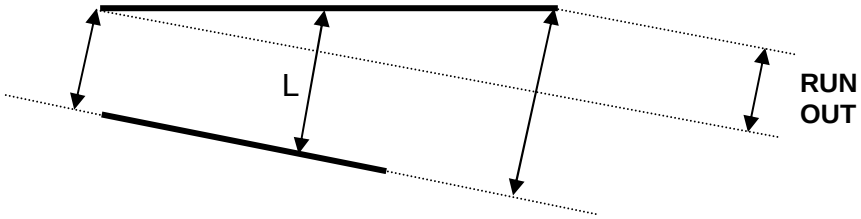


Fig. 19.7-2 Showing the separation and run out between the short line and the long line

The first line selected, in effect, becomes the reference line for the measurement. If you are measuring the distance between a long line and a short line it will be better to select the long line first (unless the short line is specifically defined as a datum).

19.8 Dimensioning to a Cone

A cone is a special case, as when measuring to it, it may be treated either as a line along it's axis, or alternatively as a point at it's apex.

Similarly, its measured diameter has little meaning (as all the measured points will be below the surface). Therefore selecting the cone twice will not display its diameter, as for circles and cylinders, but it's included angle.

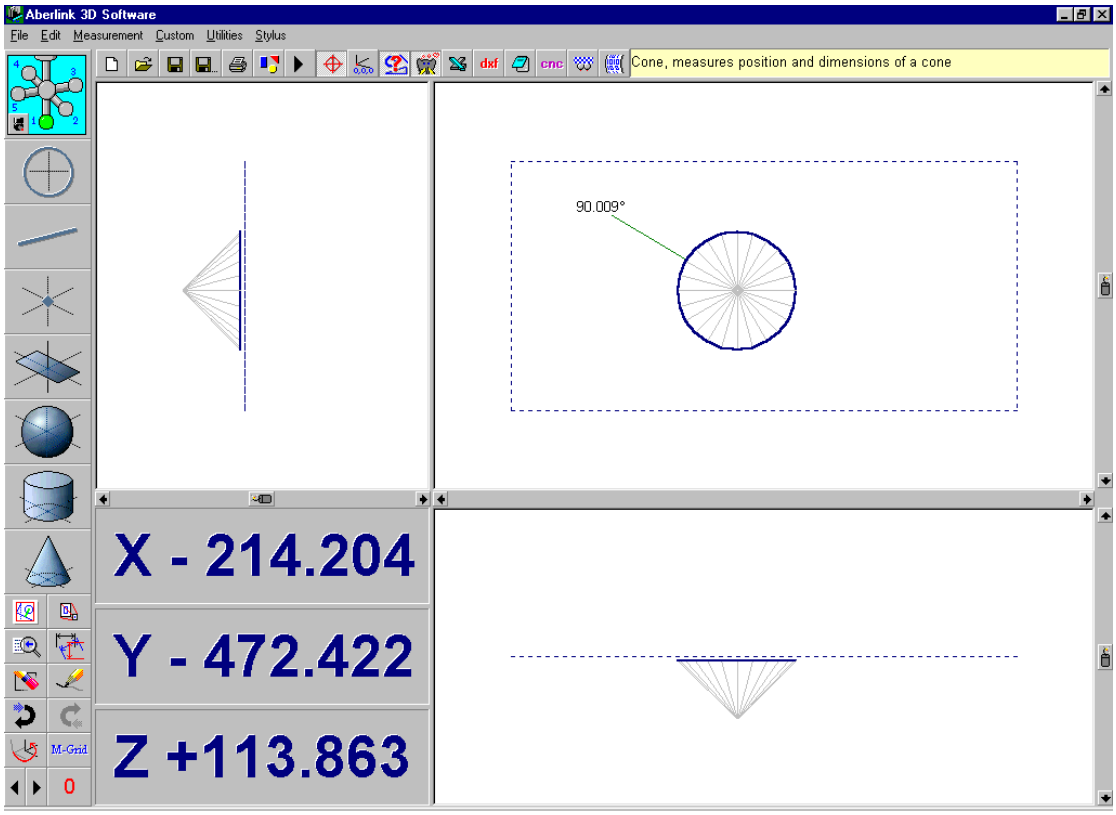


Fig.19.8-1 Showing the include angle of a cone

www.EngineeringBooksPdf.com

To measure the diameter of a cone at its end requires the construction of circle at the intersection of the cone with the plane forming its end. See chapter 12.4 – Measuring the maximum diameter of a cone.

When bringing up dimensions to a cone, the cone is treated either as a line or a point, depending on whether it is selected as the first or second feature.
If selected first, the software will treat the cone as a line along its axis, and produce a relevant dimension accordingly.

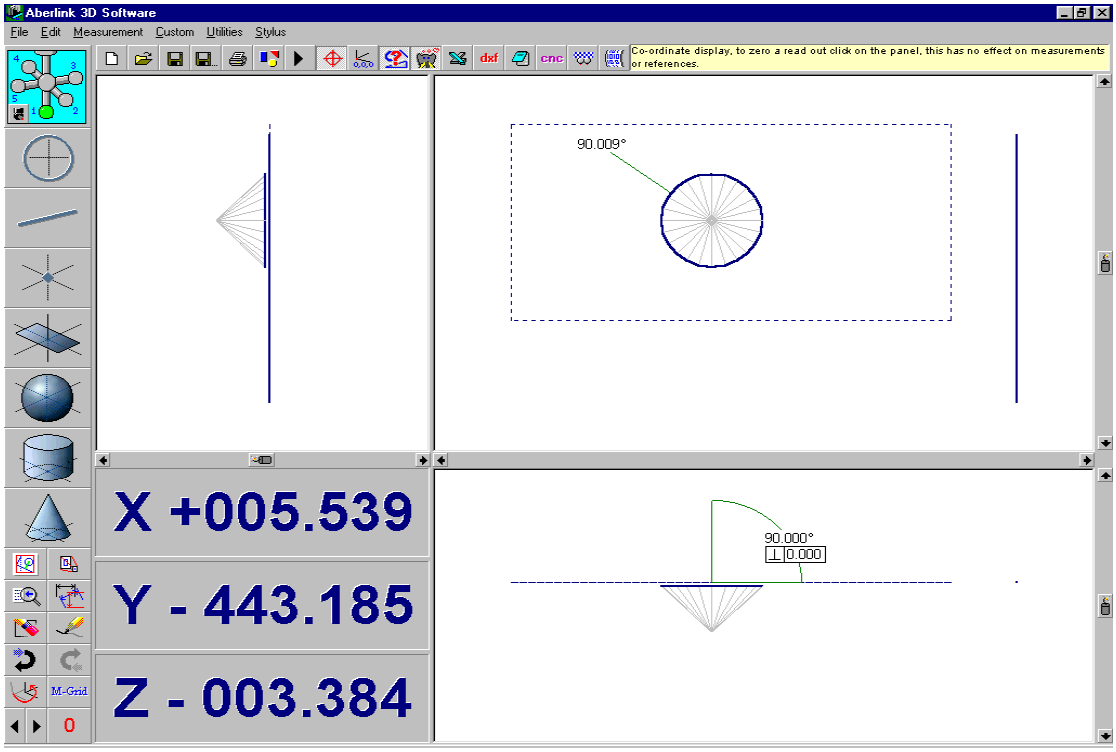


Fig. 19.8-2 Showing the cone treated as a line through its axis by selecting it first

If however, the cone is selected as the second feature, the software will treat it as a point at its apex, and again produce a relevant dimension accordingly.

www.EngineeringBooksPdf.com

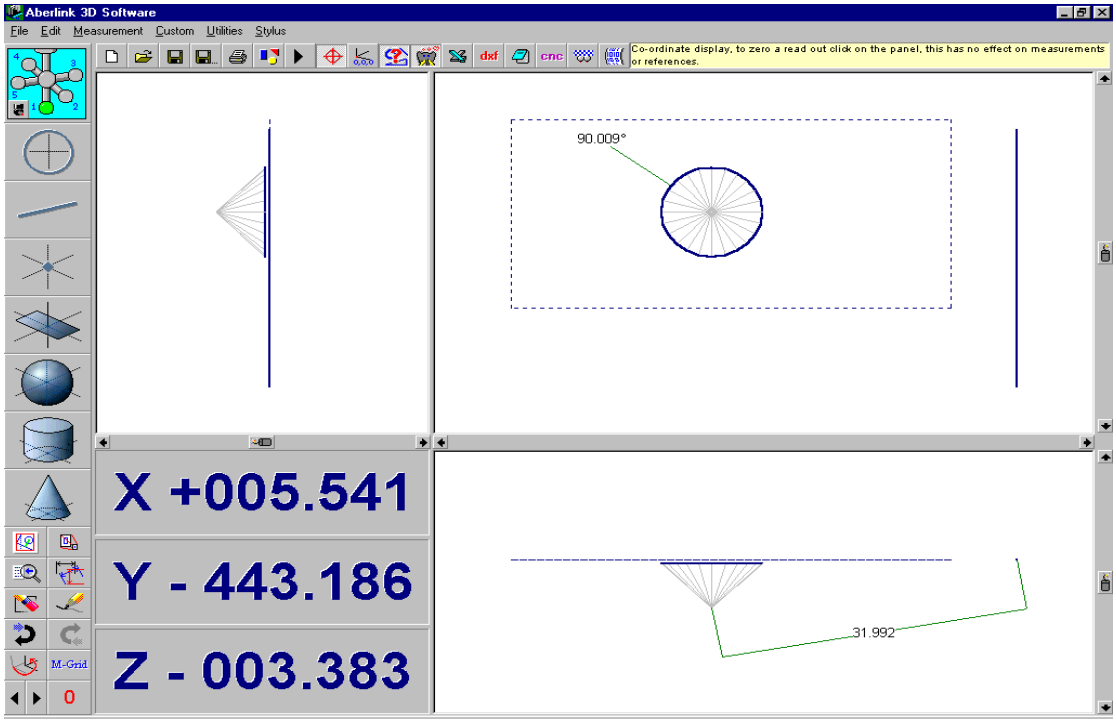


Fig. 19.8-3 Showing the cone treated as a point at its apex by selecting it second

www.EngineeringBooksPdf.com

19.9 Aligned to Feature & Ant-aligned to Feature

A dimension which is aligned or ant-aligned (perpendicular) to a feature, can be described as a dimension between two points in space, or between a line or plane and a point, in which case the dimension will be attached to the mid-point of the line or plane. Instead of being aligned between the features, these measurements will be perpendicular or parallel to the feature you have selected.

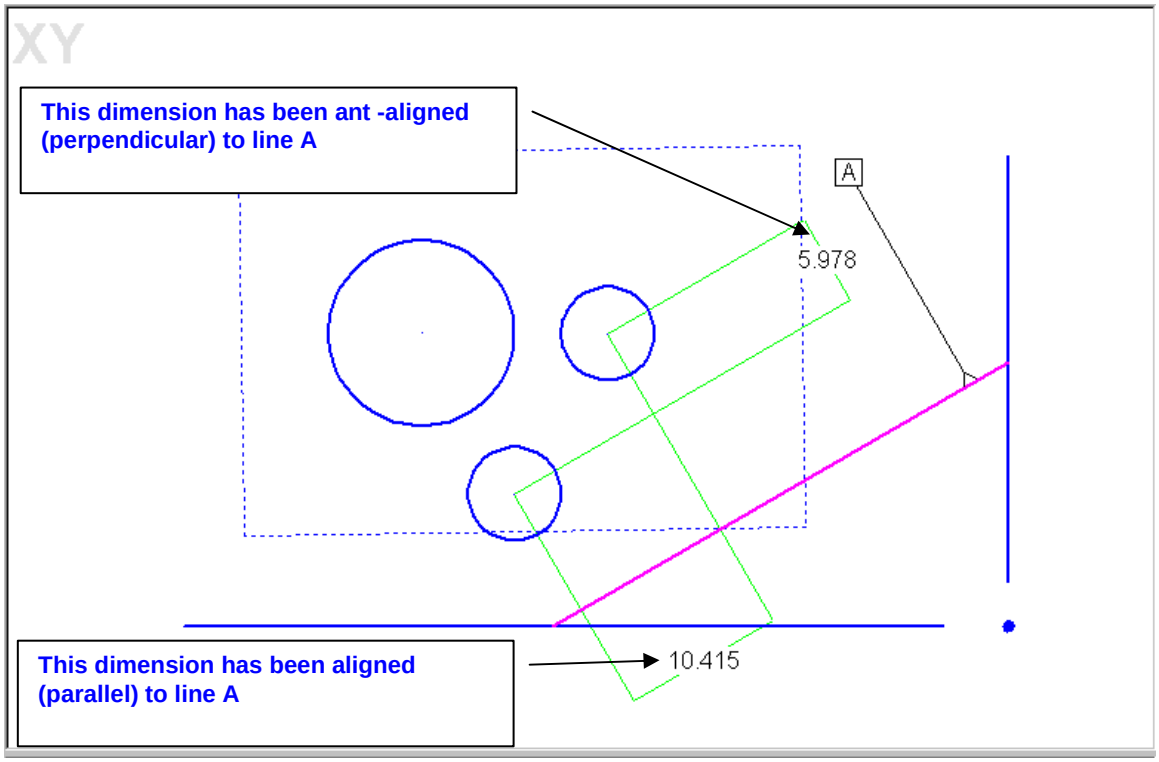


Fig. 19.8-4 Showing dimension aligned and ant-aligned to feature A

www.EngineeringBooksPdf.com

19.10 Leader option from 'Dimension type drop down menu'

If you need to know the concentricity between two selected circles or cylinders rather than the horizontal or vertical dimensions you can pick the 'Leader option' from the Dimension type drop down menu.
This will give you a geometric box snowing the concentricity between the features.

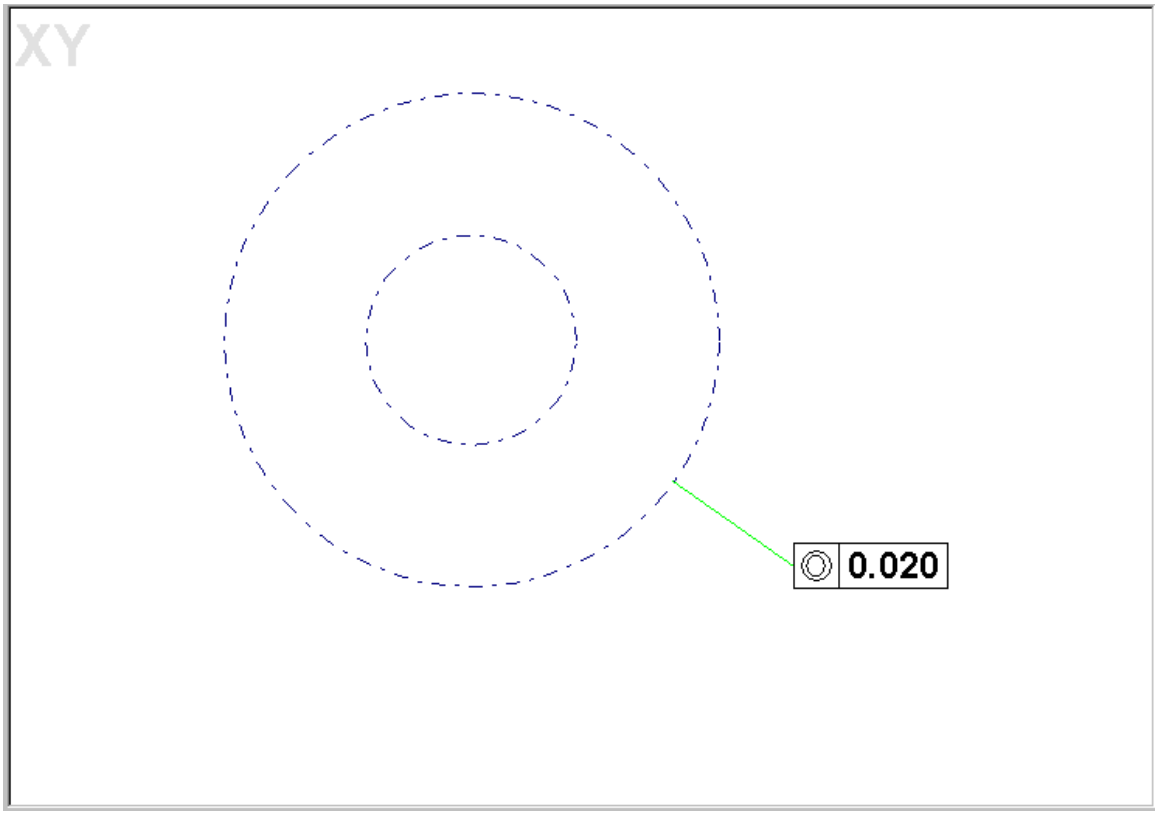
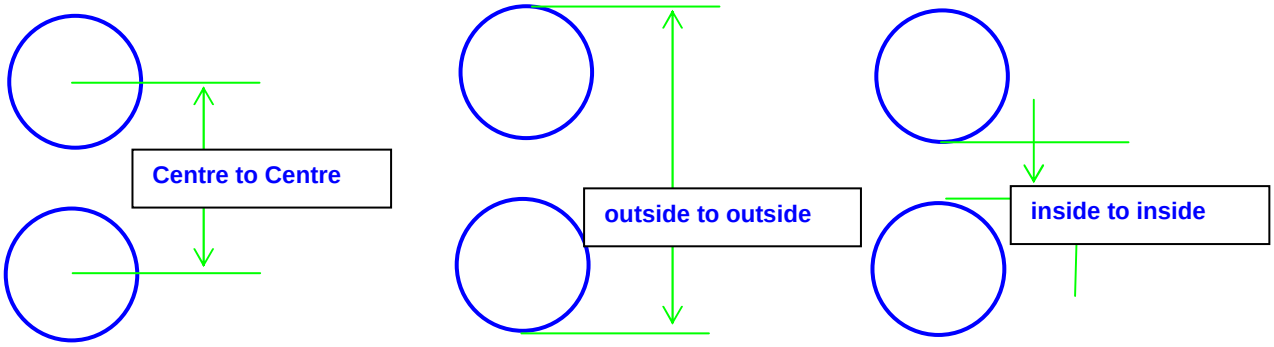


Fig. 16.8-4 Geometric Box with Leader showing concentricty

19.11 Circle Max & Min Measure Option

The standard way to measure the distant between circles is from centre to centre But you can also measure the distant between the outside of the circle (max) and the inside of the circles (min)



To do this dimension the distant between the centre of the circles as per normal

www.EngineeringBooksPdf.com

Then right hand click on the dimension to bring up the 'Dimension Details' window, at the bottom of the window there are three button marked 'centre' 'Max' & 'Min' selecting these button will changes the outputted dimensions between centre, outside & inside measurements.

19.12 Setting Nominal Values and Tolerances for Dimensions

When printing out inspection reports using Tabulated Dimensions, it is possible to print out nominal values (and hence measured error) and tolerance bands (and hence pass/fail statements). If no nominal value has been set for a dimension, then the software will use the measured value rounded to the precision defined in the software set up – see chapter 24. In this instance the tolerance band assumed will be plus or minus half the rounding value.

If however you wish to set a nominal value or tolerance band other than that defaulted to, this can be done by right mouse clicking on the relevant dimension. This will bring up the dimension details window, as follows:

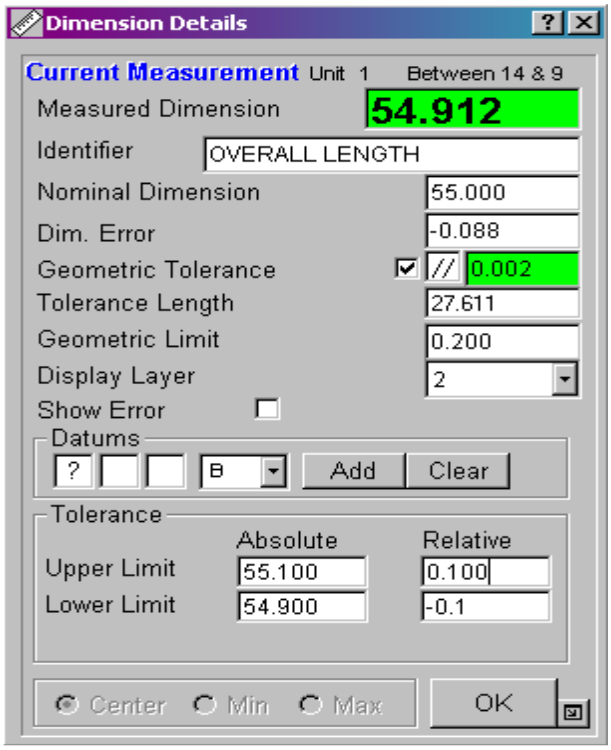


Fig. 19.9-1 The Dimension Detail window

Now the nominal dimension and tolerances can be set by simply typing over the default values that the software has selected.

Note that there is also a tick box to indicate whether you want to use this dimension when performing an optimisation. Optimisations calculate the best

www.EngineeringBooksPdf.com

alignment of the measured component to minimise the error of the selected dimensions – see chapter 16, Shifting, Rotating or Cancelling References.

Note also that if the measured dimension falls outside of the tolerance band set, then the background of the measured dimension value will become red.

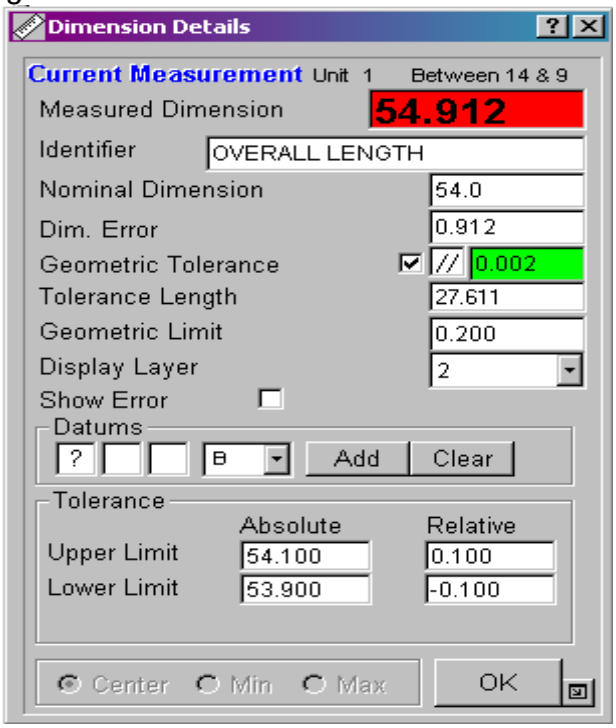


Fig. 19.9-2 The Dimension Detail window showing a measured value out of tolerance

When you OK this window and return to the main screen, the dimension leader lines will also appear red on the graphical representation.

www.EngineeringBooksPdf.com

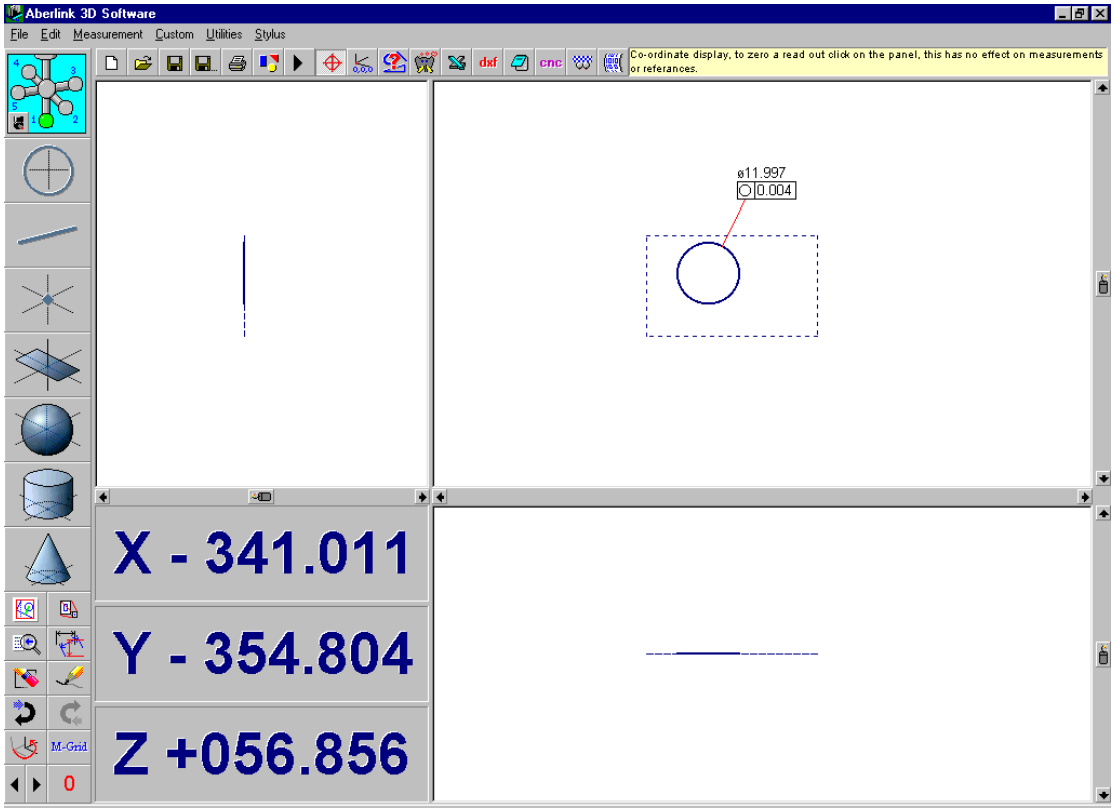


Fig. 19.9-3 Showing a graphical representation with dimension out of tolerance

Now printing out a tabulated dimensions report, it will be possible to show the nominal value, the error, the upper and lower tolerances and also a pass/fail statement.

Drawing No.	1234567	**Order No.**	900-7	**Date**	13-Feb-07 15:16
Title	Tabulated	**Serial No.**	23	**Inspector**	Chris
Customer	A.Other	**Material**	Rubber	**Notes**	

Identifier	Dimension		Dim.	Limit			Geometric Tolerance		
	Nominal	Actual	Error	Upper	Lower	Pass/Fail	Actual	Limit	Pass/Fail
OVERALL LENGTH	54.000	54.912	0.912	54.100	53.900	Fail	0.002	0.200	PASS

Fig. 19.9-4 A Tabulated dimensions inspection report

www.EngineeringBooksPdf.com

Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

Obviously in this example there will only be one dimension contained in the report. For more information on tabulated dimension reports see chapter 23 – Printing Results.

www.EngineeringBooksPdf.com

20.0 GEOMETRICAL TOLERANCES

Interdiction to Geometric Dimensioning and Tolerancing
Geometric Dimensioning and Tolerancing (GD&T) is a universal language of symbols, much like the international system of road signs that advise drivers how to navigate the roads. GD&T symbols allow a Design Engineer to precisely and logically describe part features in a way they can be accurately manufactured and inspected. GD&T is expressed in the feature control frame (Figure 17.0-0). The feature control frame is like a basic sentence that can be read from left to right. For example, the feature control frame illustrated would read: The 5 mm square shape (1) is controlled with an all-around (2) profile tolerance (3) of 0.05 mm (4), in relationship to primary datum A (5) and secondary datum B (6). The shape and tolerance determine the limits of production variability.

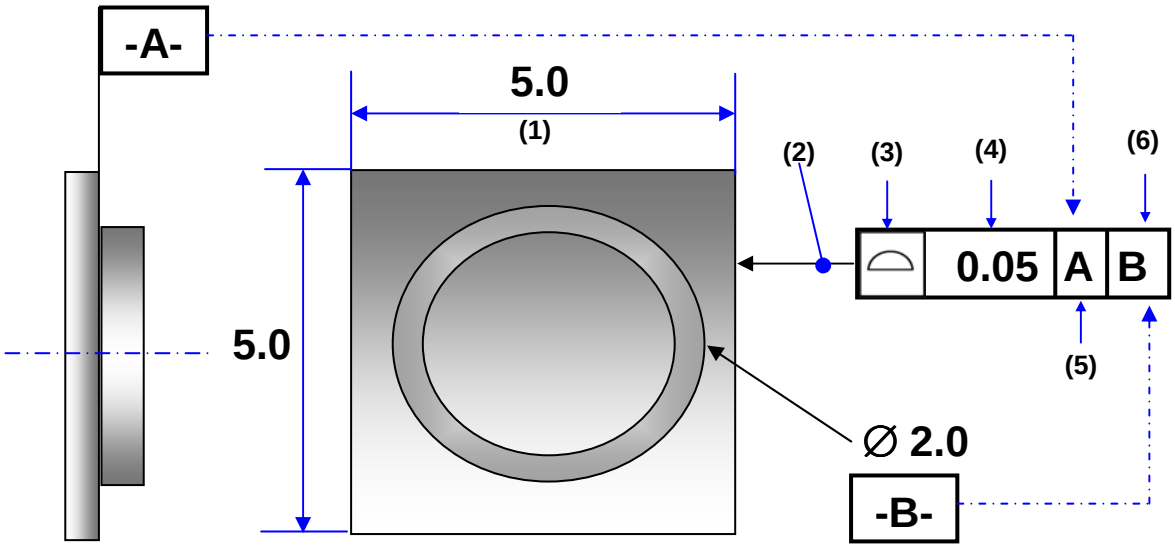


Fig. 20.0-0

There are seven shapes, called geometric elements, used to define a part and its features. The shapes are: point, line, plane, circle, cylinder, cone and sphere. There are also certain geometric characteristics that determine the condition of parts and the relationship of features.

Where relevant the geometric tolerance applicable to each measurement unit (eg. Roundness of circles, straightness of lines) will be displayed underneath the graphical representation of that unit.

www.EngineeringBooksPdf.com

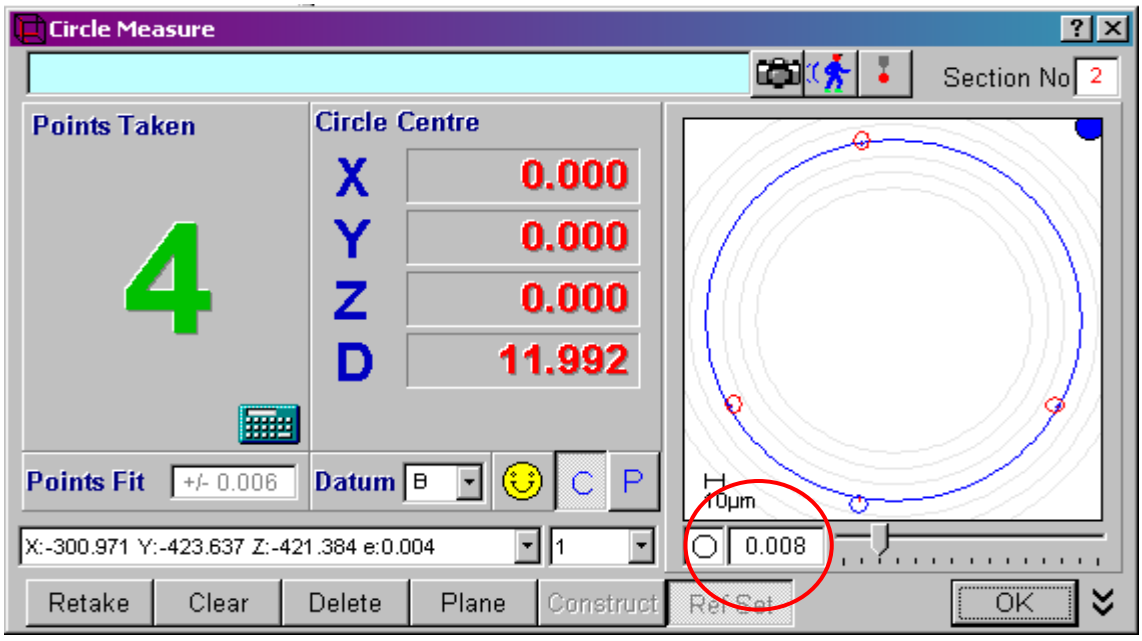


Fig. 20.0-1 Graphical representation of a circle in the Circle Measure window

Additionally the True Position and Maximum Material Condition for circles may be obtained from the 'Dimension Details Window'.

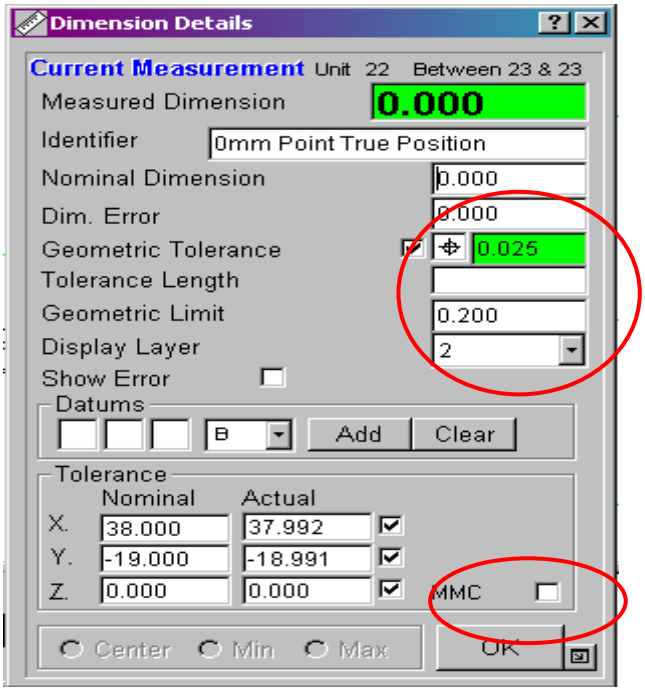


Fig. 20.0-2 A Circle Measure window showing additional information

None of these geometric tolerance values will by default be displayed on the graphical representation of the component that is built up during the inspection.

www.EngineeringBooksPdf.com

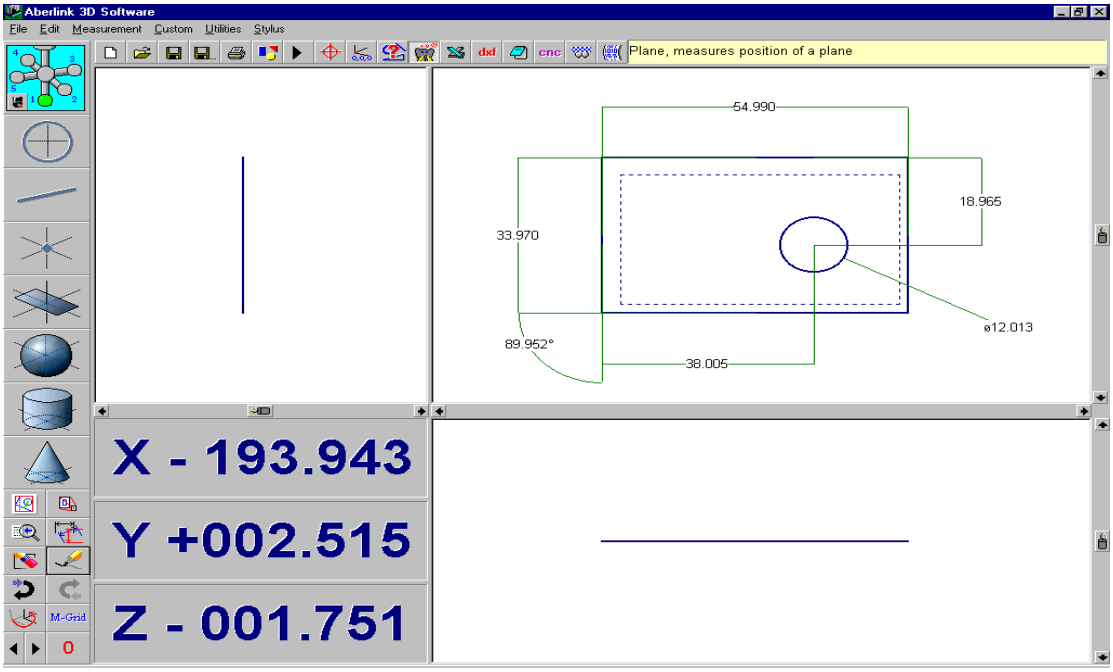


Fig. 20.0-3 Dimensions displayed without Geometric Tolerances

To display on the screen all geometric tolerance values relevant to the inspection simply click on the Geometric Tolerance button.

Fig. 20.0-4 The Geometric Tolerance button

The geometric tolerance values that are relevant to the measurements on the screen will now be displayed.

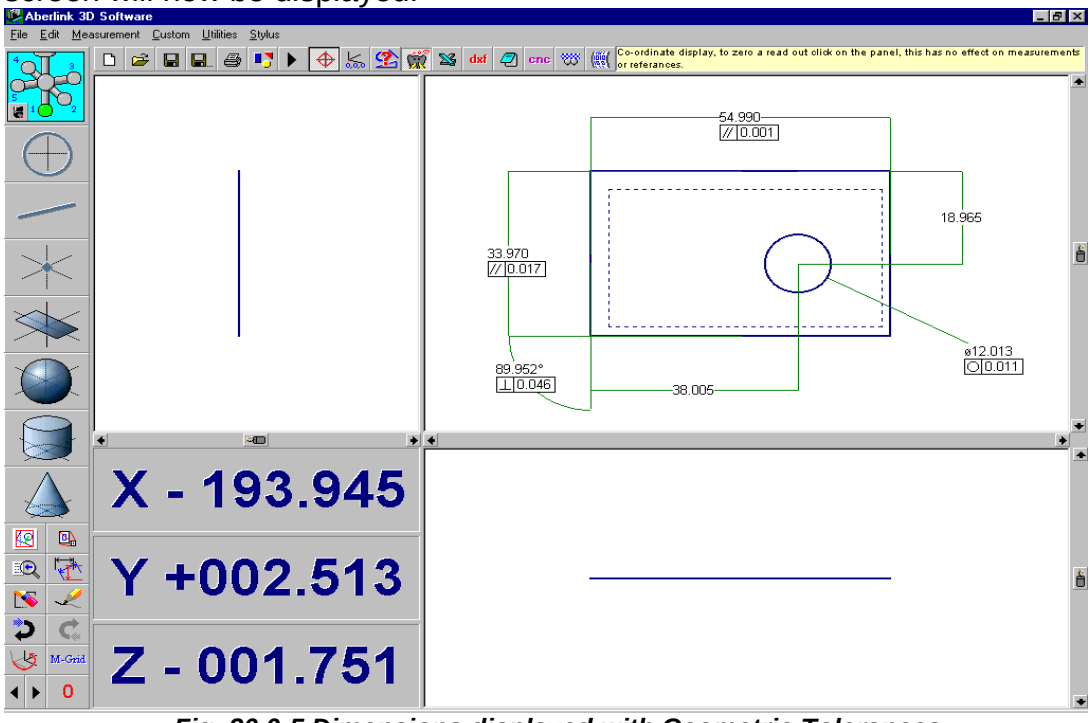


Fig. 20.0-5 Dimensions displayed with Geometric Tolerances

www.EngineeringBooksPdf.com

Now to display the geometric tolerances that are relevant to any particular feature, (for instance flatness of a plane, straightness of a line) click twice on that feature and the outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and click for a third time. The geometric tolerance will now appear on the screen.

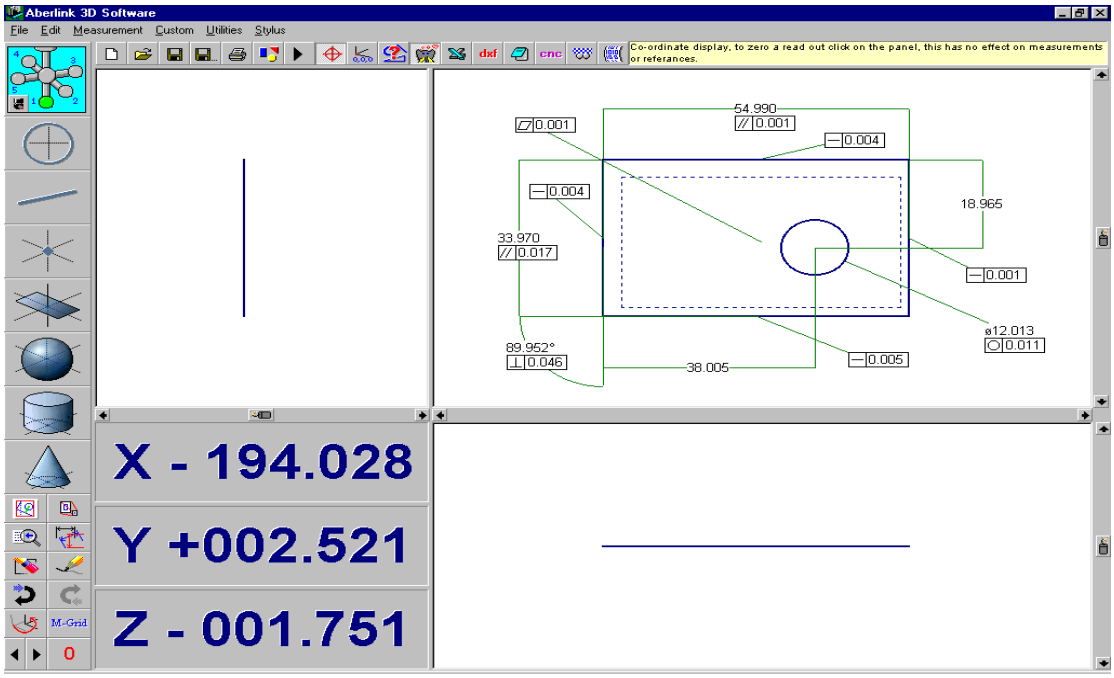
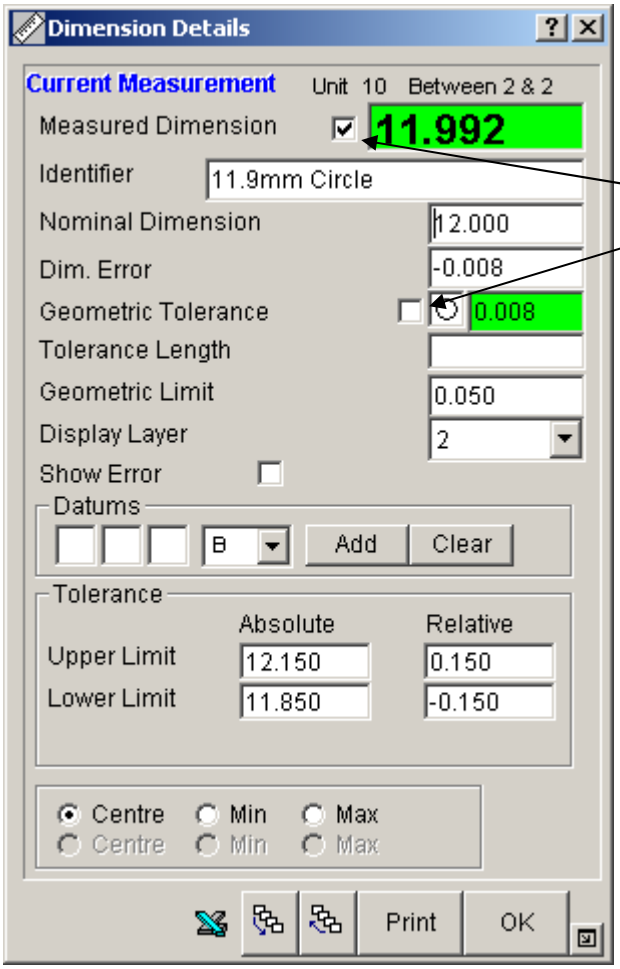


Fig. 20.-6 Geometric Tolerances displayed for specific features

It is also possible to choose to display either the standard values like diameter, distances and angles or the geometric values like roundness, parallelism and angularity.

This is useful feature because some reports like multiple components and SPC reports only record one value for each measured element.
For example if wanted to record the diameter and the roundness of a circle in an SPC report you would need to dimension to circle twice the 1st time you would select the display standard value (diameter) option, the 2nd time you dimension the circle would select the display geometric values (roundness) option now both values will be display in the report.

www.EngineeringBooksPdf.com



If both of these boxes are tick both values will be displayed if you un-tick the geometric tolerance box only the measured values will be displayed and vice versa if you un-tick the measured dimension box and leave the geometric tolerance box ticked only the geometric values will be displayed.

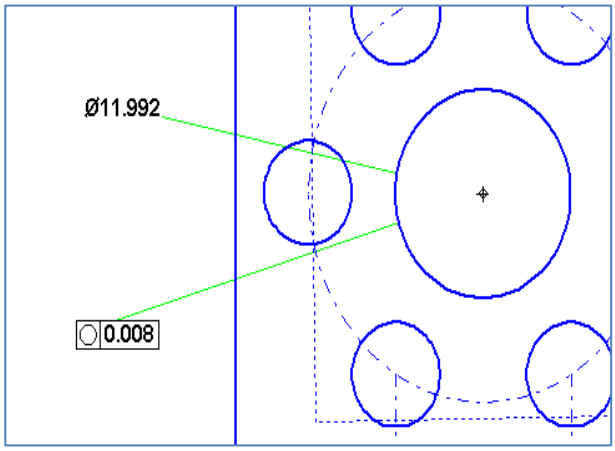
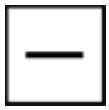


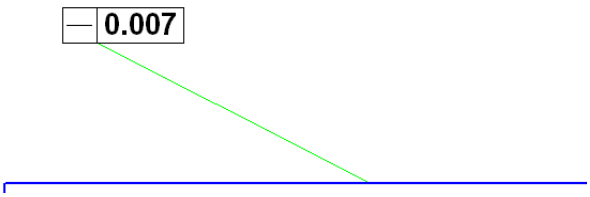
Fig. 20.-7 Switching between Geometric Tolerances and Measured Dimension

Geometric Characteristic Symbols

20.1 Straightness



— A condition where all points are in a straight line, the tolerance specified by a zone formed by two parallel lines
To display the straightness of a line click twice on the line and the outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and click for a third time. The geometric tolerance will now appear on the screen.

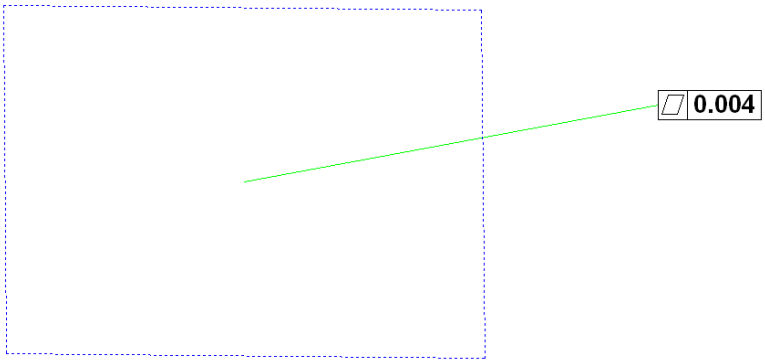


20.2 Flatness

— All the points on a surface are in one plane, the tolerance specified by a zone formed by two parallel planes.
To display the flatness of a plane click twice on the plane and the

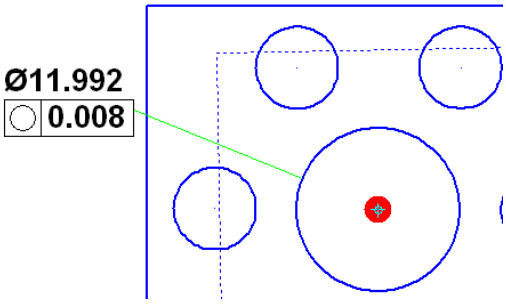
www.EngineeringBooksPdf.com

outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and click for a third time. The geometric tolerance will now appear on the screen



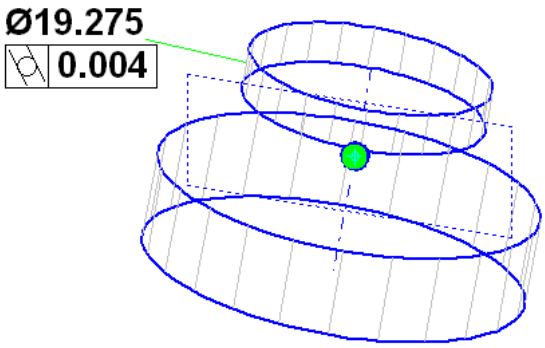
20.3 Roundness or Circularity

— All the points on a surface are in a circle. The tolerance is specified by a zone bounded by two concentric circles
If the geometric tolerance button is switch on, the roundness of a circle will be displayed automatically underneath the diameter in the graphic window.



20.4 Cylindricity

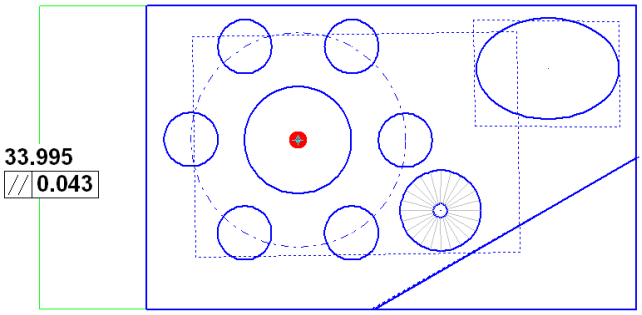
— All the points of a surface of revolution are equidistant from a common axis. A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders within which the surface must lie.
If the geometric tolerance button is switch on, the cylindricity of a cylinder will be displayed automatically underneath the diameter in the graphic window.



www.EngineeringBooksPdf.com

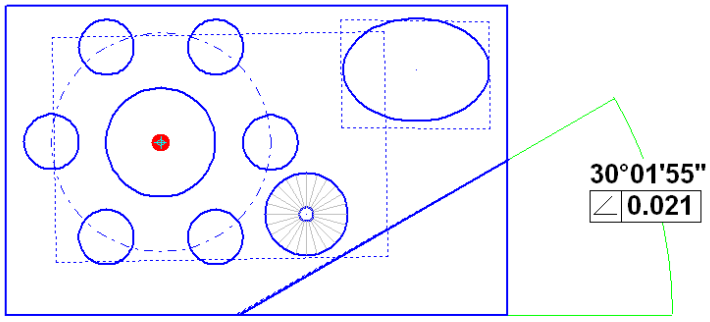
20.5 Parallelism

— The condition of a surface or axis equidistant at all points from a datum plane or axis. Parallelism tolerance specifies one of the following: a zone defined by two planes or lines parallel to a datum plane or axis, or a cylindrical tolerance zone whose axis is parallel to a datum axis.
If the geometric tolerance button is switch on, the parallelism of between two lines or planes will be displayed automatically underneath the dimension in the graphic window.



20.6 Angularity

— The condition of a surface or axis at a specified angle (other than 90°) from a datum plane or axis. The tolerance zon e is defined by

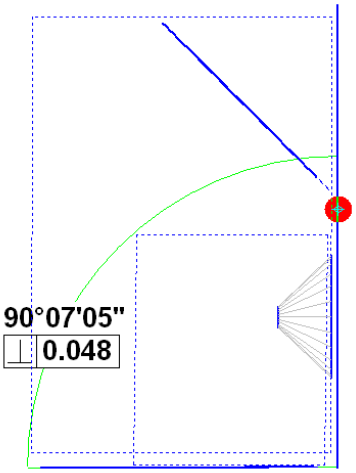


two parallel planes at the specified basic angle from a datum plane or axis.
If the geometric tolerance button is switch on, the angularity of between two lines or planes will be displayed automatically underneath the angle in the graphic window.

20.7 Perpendicular (Squareness)

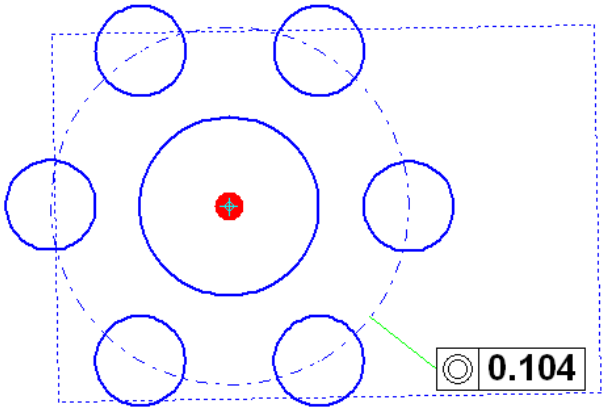
— The condition of a surface or axis at a right angle to a datum plane or axis. Perpendicularity tolerance specifies one of the following: a zone defined by two planes perpendicular to a datum plane or axis, or a zone defined by two parallel planes perpendicular to the datum axis.
If the angle is between 96° 84° and geometric tolerance button is switch on, the perpendicular (squareness) between two lines or planes will be displayed automatically underneath the angle in the graphic window.

www.EngineeringBooksPdf.com



20.7 Concentricity

— The axes of all cross sectional elements of a surface of revolution are common to the axis of the datum feature. Concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with the datum axis.
To display the concentricty of two circles click on the first and then click on the



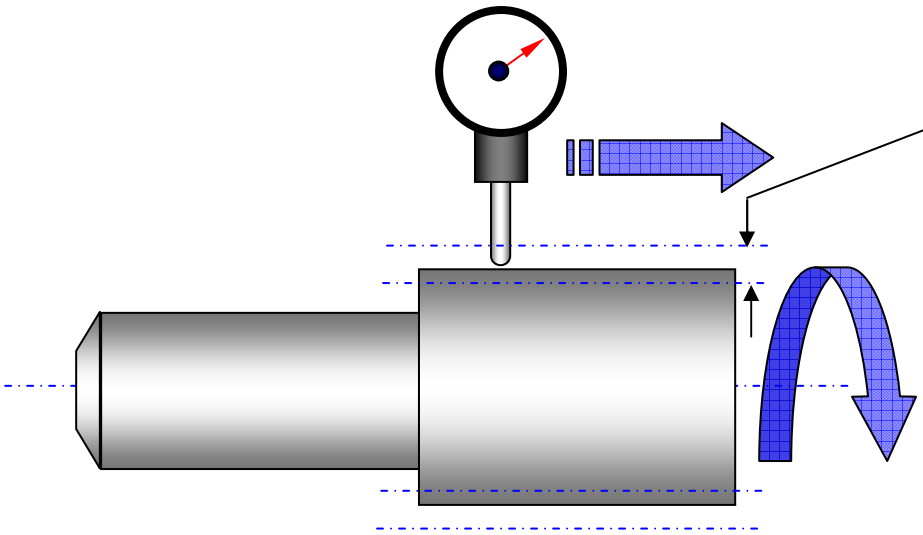
second after this outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and click for a third time. The geometric tolerance will now appear on the screen.

20.8 Total Runout

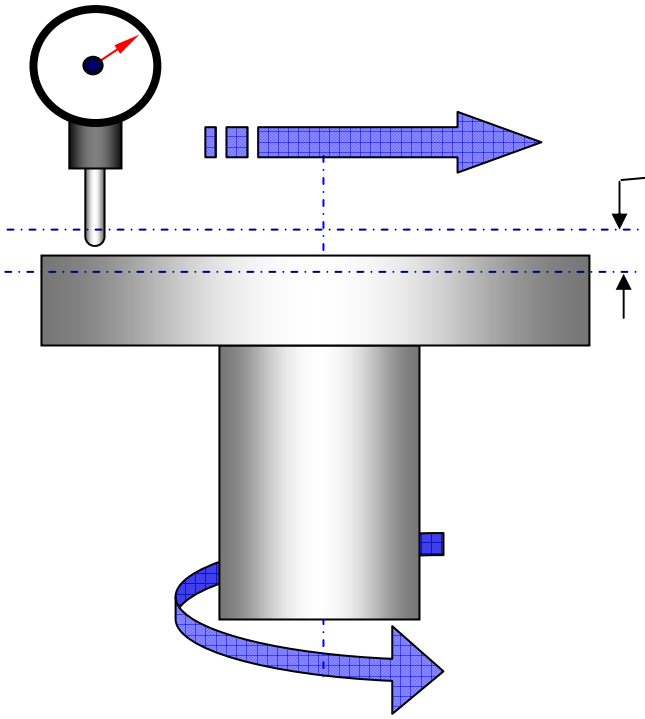
— Provides composite control of all surface elements. The tolerance applied simultaneously to circular and longitudinal elements as the part is rotated 360 degrees. Total runout controls cumulative variation of circularity, cylindricity, straightness, coaxiality, angularity, taper, and profile when it is applied to surfaces constructed around a datum axis. When it is applied to surfaces constructed at right angles to a datum axis, it controls cumulative variations of perpendicularity and flatness.

www.EngineeringBooksPdf.com

In Aberlink 3D its possible to display the Total Runout between two cylinders / lines or between an cylinder / line and a plane perpendicular to the cylinder.



The cylinders surface must lie within the specified runout tolerance zone (0.001 full indicator movement) when the part is rotated 360° about the datum axis anywhere along the cylinders length.

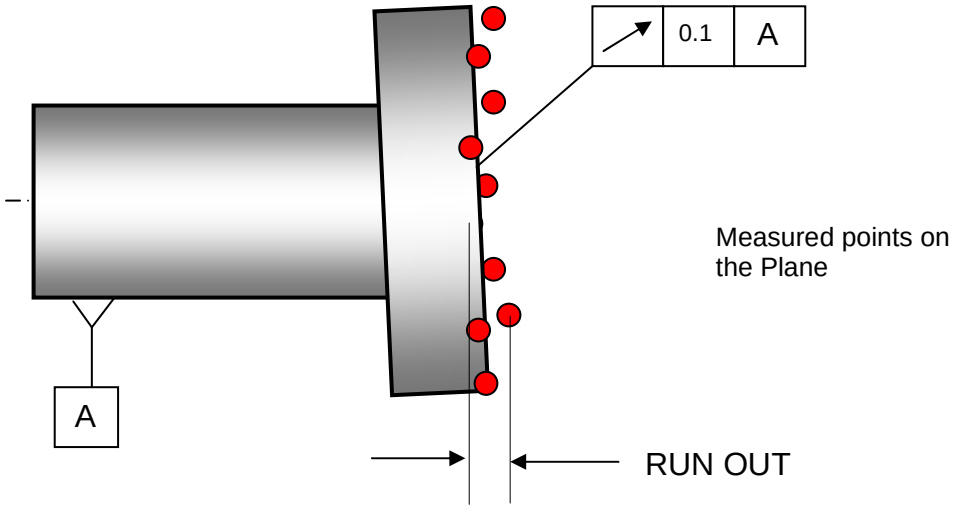


The top of the cylinders must lie within the specified runout tolerance zone (0.001 full indicator movement) when the part is rotated 360° about the datum axis anywhere along the top of the cylinder.

www.EngineeringBooksPdf.com

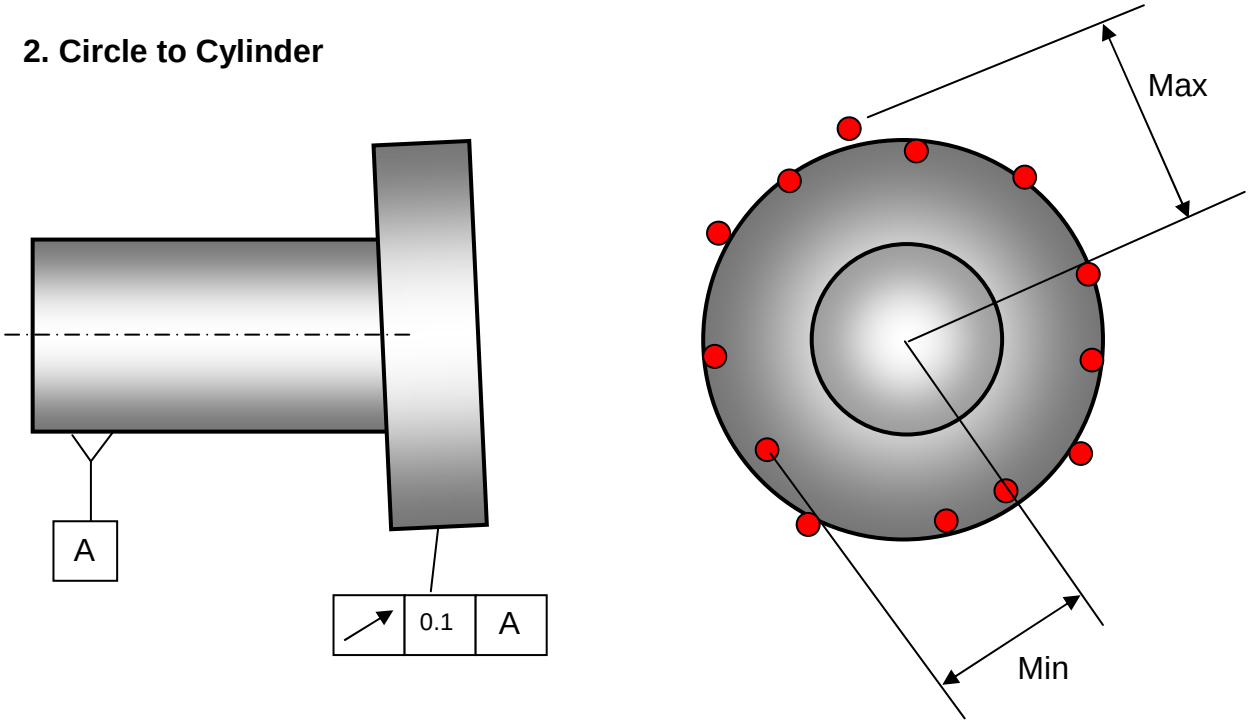
20.8.1 Run out.

1. Plane to Cylinder



Run Out between Cylinder (datum A) and a Plane will be defined as the maximum variation between the measured points on the plane in the direction parallel to the axis of the cylinder.
To call up the run out click on the Cylinder first and then the Plane second.
If the Plane is selected first, then the squareness between the features will be reported instead of Run Out

2. Circle to Cylinder



Run Out between Cylinder (datum A) and a Circle will be defined as the maximum variation between the measured points on the Circle in the direction perpendicular to the axis of the cylinder (Max – Min).

www.EngineeringBooksPdf.com

To call up the run out of the Circle relative to the Cylinder click on the Cylinder first and then the Circle second.

If the Circle is selected first, then the Total Run Out of the Cylinder relative to the circle will be reported. Again this will use the maximum variation in the measured points on the cylinder relative to the Axis of the Circle, which will be defined by whichever plane the circle is projected into.

Total Run Out will apply exactly the same principles and will be available for Cylinder to Cylinder measurements or Cylinder to Circle measurements, as detailed above.

20.9 True Position

— A positional tolerance defines a zone in which the center axis or centre plane is permitted to vary from true (theoretically exact) position. Basic dimensions establish the true position from datum features and between interrelated features. A positional tolerance is the total permissible variation in location of a feature about its exact location. For cylindrical features such as holes and outside diameters, the positional tolerance is generally the diameter of the tolerance zone in which the axis of the feature must lie. For features that are not round, such as slots and tabs, the positional tolerance is the total width of the tolerance zone in which the centre plane of the feature must lie.
To display the True Position of a point or circle L/H click twice on that point or circle then R/H click you will see a drop down menu pick ‘Cartesian / Polar Position’ option and the outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and L/H click for a third time. The geometric tolerance will now appear on the screen.
It is possible to allocate datum letters to the features after you have measure them this is useful if wish to match the datum letters marked on the draw to the feature you have measured.

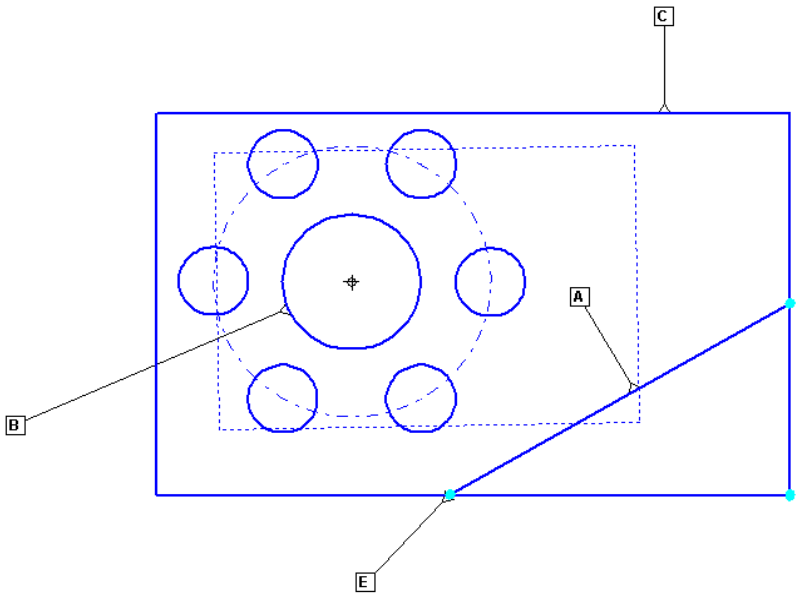


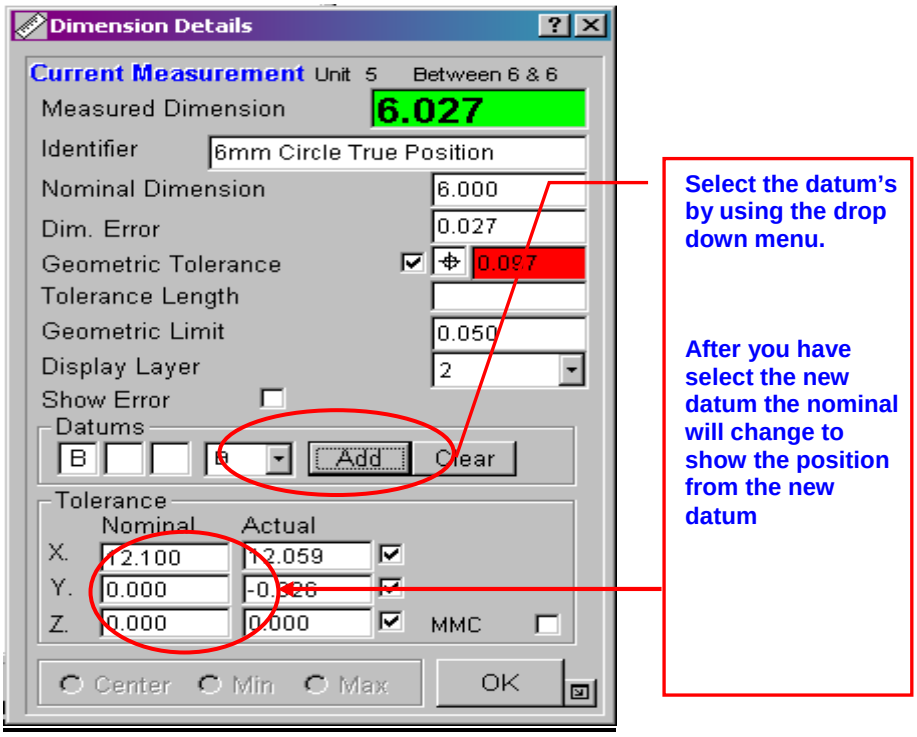
Fig. 200-7 Datum Letter displayed for specific features

www.EngineeringBooksPdf.com

To allocate a datum letter to a feature 'right click' on that feature with the mouse this will open the 'Measurement Window' you can now use the drop menu next to the word 'Datum' to select the letter you wish to assign to that feature.
It is possible to display the datum letters in the graphic windows by left hand click on a feature twice then 'right hand click' the mouse you will get a drop down menu select the 'datum option' from the menu then left hand click on the screen to display the datum letter.

20.9.1 True Position relative to specified datum(s)

If the true position that you are measuring is relative to some other point on the component other than that which has been set as 0,0,0 then it is possible to show it relative to other specified datums. For instance if you have chosen a large hole as the reference (because it is an easy feature to measure automatically) but need to show the true position of a hole relative to a smaller hole specified as datum B, then this can also be done in the dimension details window:
Note:- **to display the true position of a circle or a point from features other than the origin you must to allocate datum letters to those features .**



Select datum B and click on the 'Add' button. Also make sure that the nominal position of the hole is now relative to this datum. When you now return to the main screen the true position will now be displayed relative to datum B:

www.EngineeringBooksPdf.com

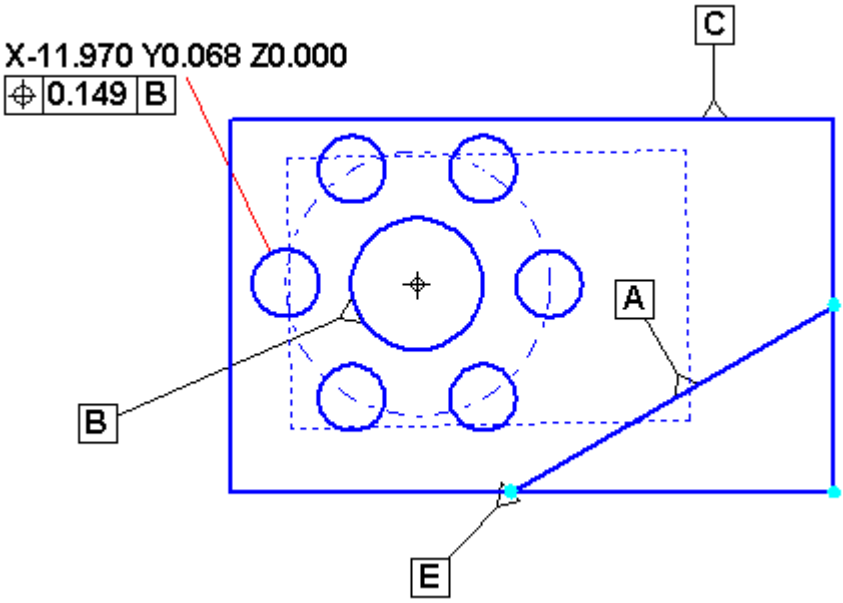


Fig. 20.0-8 True position of circle from datum B

It is possible to select up to 3 datums relative to a true position:

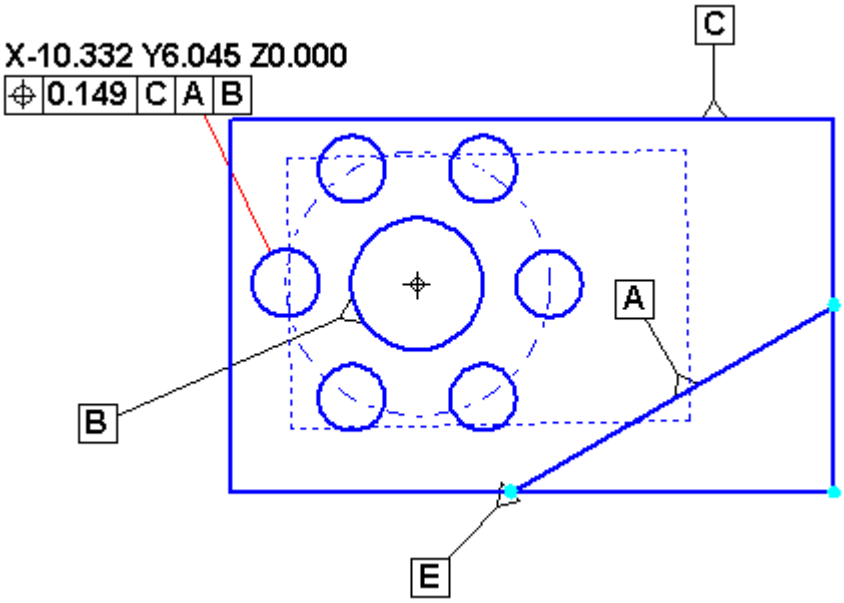


Fig. 20.0-9 True position of circle from datum C,A & B

The software will not automatically know the correct position for the circle and will just have rounded the measured values to achieve a ‘guess’ at its position. You must therefore check that the correct position values have been entered. Right click on the green dimension line attached to the position information to bring up the dimension details window:

www.EngineeringBooksPdf.com

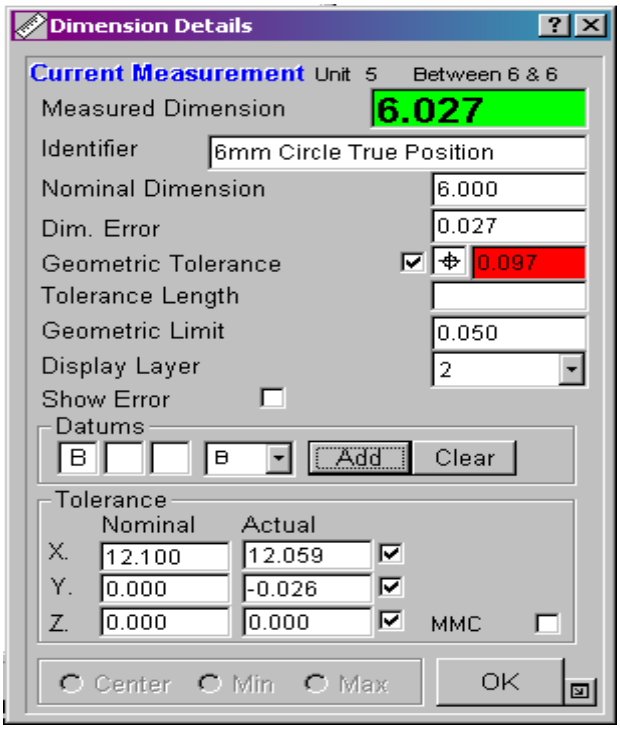


Fig. 20.0-10 True position of circle from datum B

The position of the circle is at the bottom of the details section. If you change these nominal values the true position value will change accordingly. Note that these values are relative to the datum position on the component being measured.

20.9.2 Maximum Material Condition

The maximum material condition for a hole can also be displayed if required. Again in the dimension details window tick the ‘mmc’ box. This will make two further boxes appear. In the top box type the minimum allowable diameter for the hole. The lower box will then be automatically filled in with the ‘bonus’ tolerance due to the ‘mmc’:

www.EngineeringBooksPdf.com

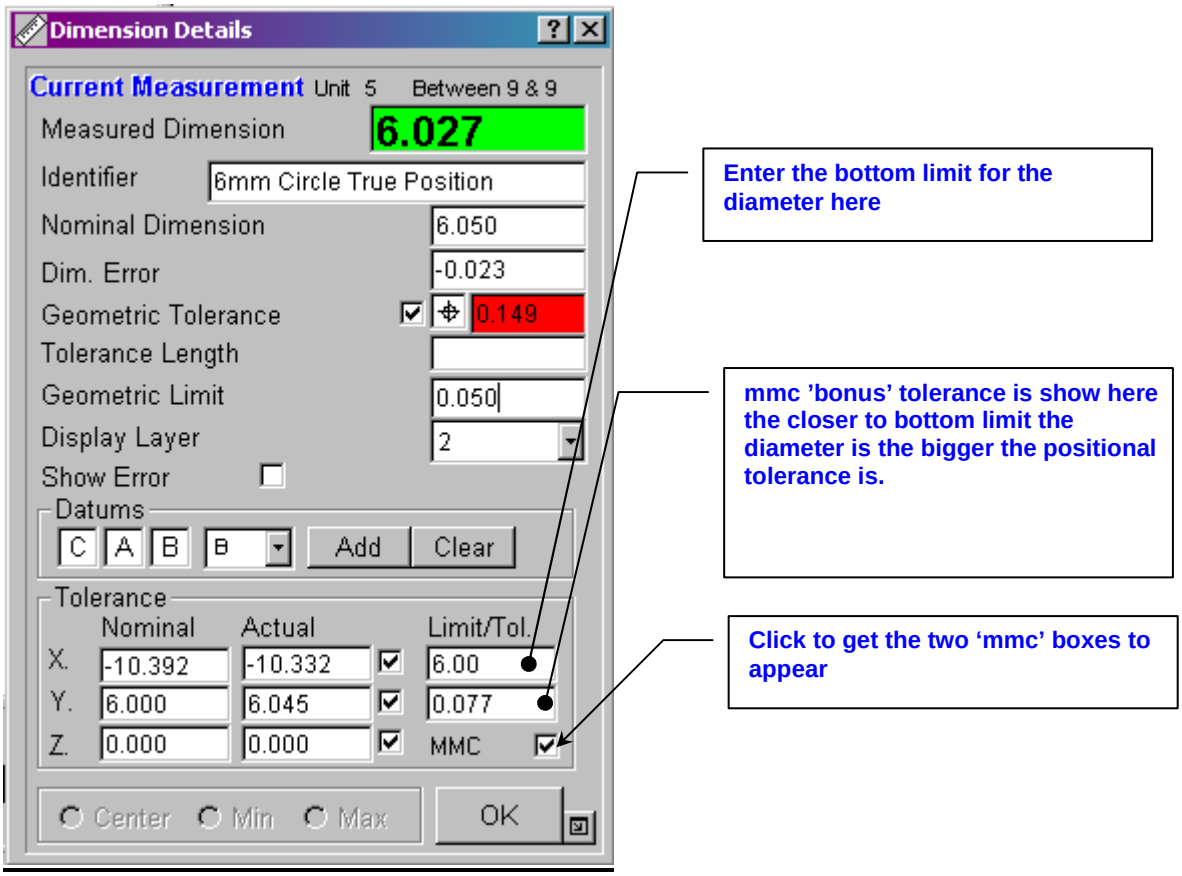
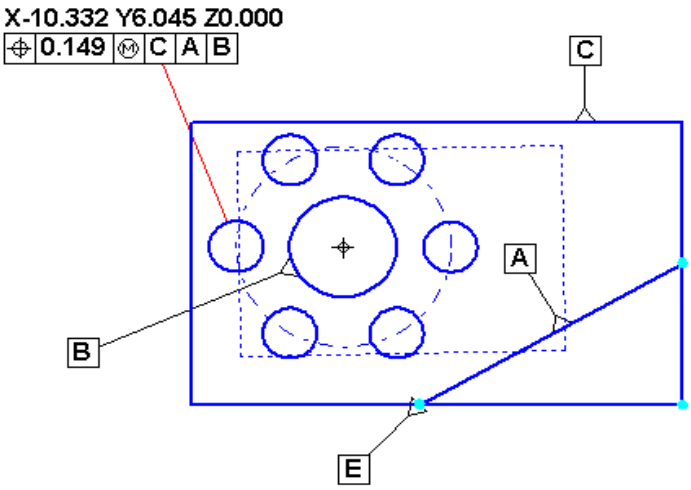


Fig. 20.0-11 True position of circle from datum B with 'MMC'

When you return to the main screen the 'mmc' symbol will be present and the



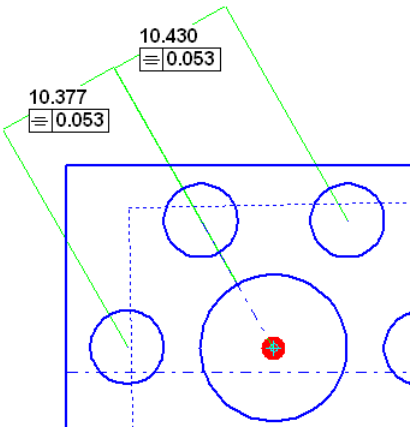
true position value will be adjusted by the bonus tolerance:

www.EngineeringBooksPdf.com

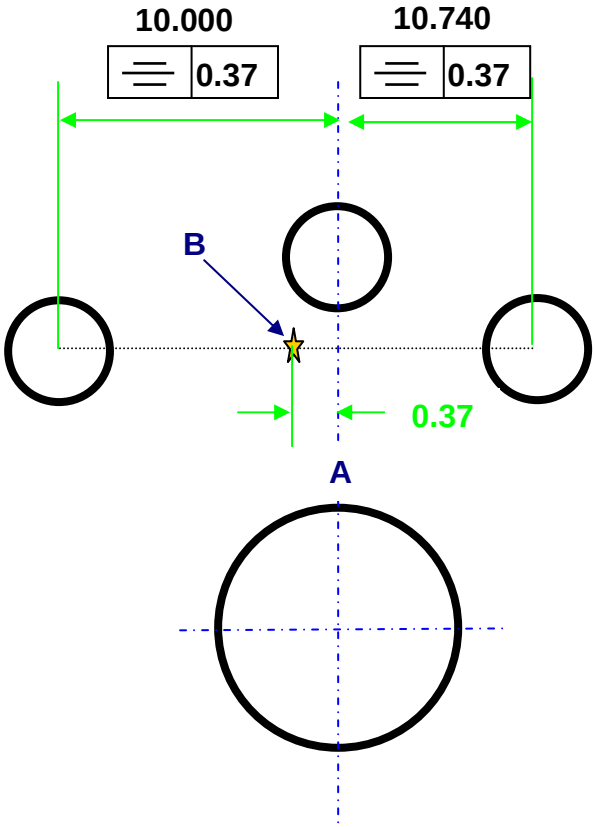
20.10 Symmetry

- The condition where the median points of all opposed elements of two or more feature surfaces are congruent with the axis or centre plane of a datum feature.

If the geometric tolerance button is switch on, the symmetry between two features and a centre line will be displayed automatically underneath each dimension in the graphic window



In Aberlink 3D the symmetry result is calculated from three features the centre line feature **(A)**, and the mid point **(B)** between two other features the difference between the mid point and the centre line feature is displayed as symmetry under both dimension in the graphic window.

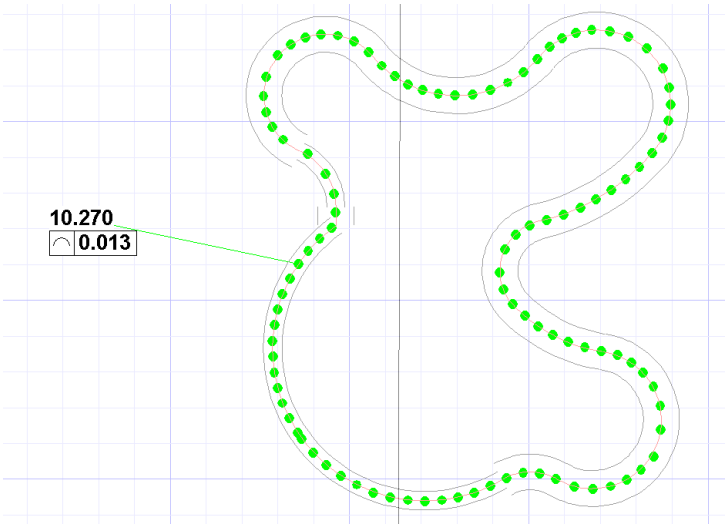


www.EngineeringBooksPdf.com

20.11 Profile

— A Tolerancing method of controlling irregular surfaces, lines, arcs, or normal planes. Profiles can be applied to individual line elements or the entire surface of a part. The profile tolerance specifies a uniform boundary along the true profile within which the elements of the surface must lie.

If you best-fit a DXF file to a measured curve it is possible to display the profile tolerance by L/H click twice on the curve If you than R/H click you will see a drop down menu pick ‘Profile’ option and the outline of a leader line will appear. Move the cursor to a convenient position on the screen, where you would like to site the dimension, and L/H click for a third time. The geometric tolerance will now appear on the screen.



www.EngineeringBooksPdf.com

21.0 GENERAL INFORMATION ABOUT MEASURED UNITS

21.1 Recalling a measured Unit

When you 'OK or 'Set Ref' a measurement unit, and the measurement window disappears and the measured feature is displayed on the main screen, it is not too late to modify the unit by adding or deleting measurement points, or changing its reference status.

Each measured unit can be recalled by right mouse clicking on the graphical representation of the feature in the main screen.

Having recalled a measurement unit it can be modified in exactly the same way as when first being measured. Hence the 'Retake', 'Clear' and 'Delete' buttons are all functional, or alternatively extra measurement points may be taken on the feature. When you have completed the required modifications the 'OK' and 'Set Ref' plus 'OK'buttons apply as usual.

21.2 Labelling Specific Measurement Units

The light blue bar at the top of every measurement unit is designed to allow text to be typed in it, in order to label each specific measurement unit. This may be useful if an inspection consists of similar or close features.

To insert text, simply click on the bar, and then enter the text via the keyboard.

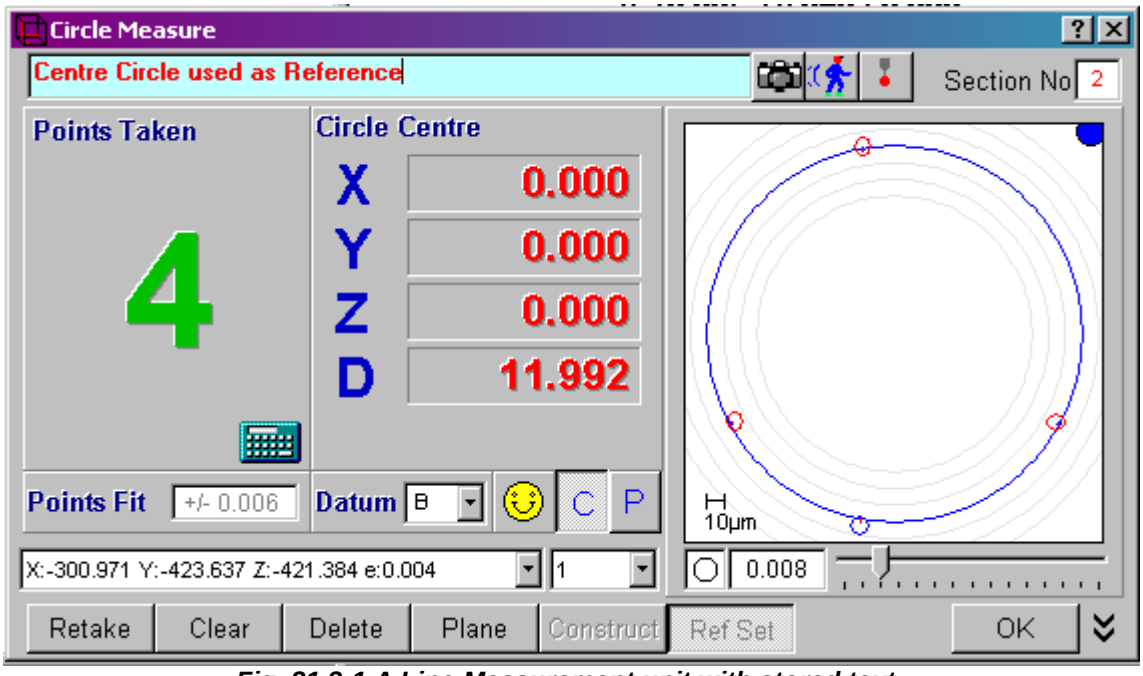


Fig. 21.2-1 A Line Measurement unit with stored text

This may be particularly useful when using the Play function – see chapter 25

www.EngineeringBooksPdf.com

21.3 Counting Down

By default all measurement units (except cones when the axis cannot be defined) will display Zero when called up, and then start counting upwards as measurement points are taken.

However if desired, there is an option within the Software Set up (see chapter 24) to select counting down when measuring features. If selected, each type of measurement unit will start at the number defined in the Set up, and then start counting down as measurement points are taken

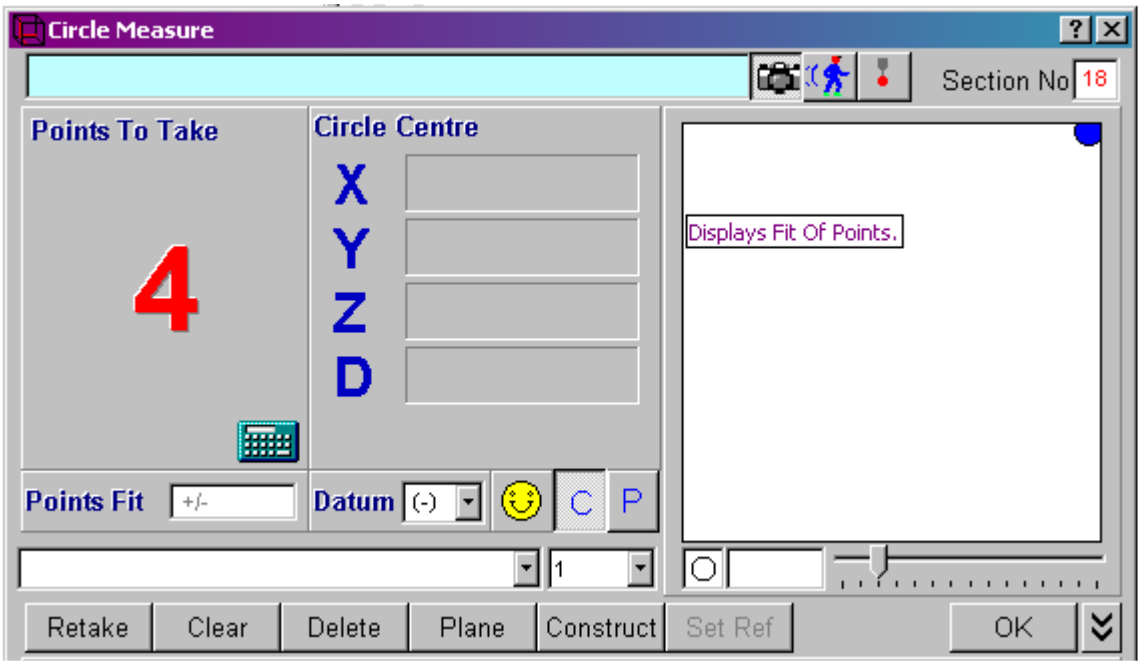


Fig. 21.3-1 A Circle Measure window ready to count down 4 points

Note that no points have yet been taken, but the large number shows 4 points to take. When 4 measurement points have been taken the window will automatically close, and the software will return to the main screen.

When measuring a plane, a line or a circle in this way, the software will automatically reference the feature if no similar feature has previously been referenced. If a similar feature has been referenced then the software will simply OK the new feature.

This function will speed up inspections by reducing the need to close measurement windows, but takes away the flexibility of the operator deciding how many measurement points are necessary to adequately inspect a feature.
Again this function may be particularly useful when using the Play function – see chapter 25. When this function is in operation the play function will both automatically open and close all measurement windows. It is worth noting that when used in conjunction with the play function, that features will be automatically referenced if previously referenced, and OK'd if previously OK'd

www.EngineeringBooksPdf.com

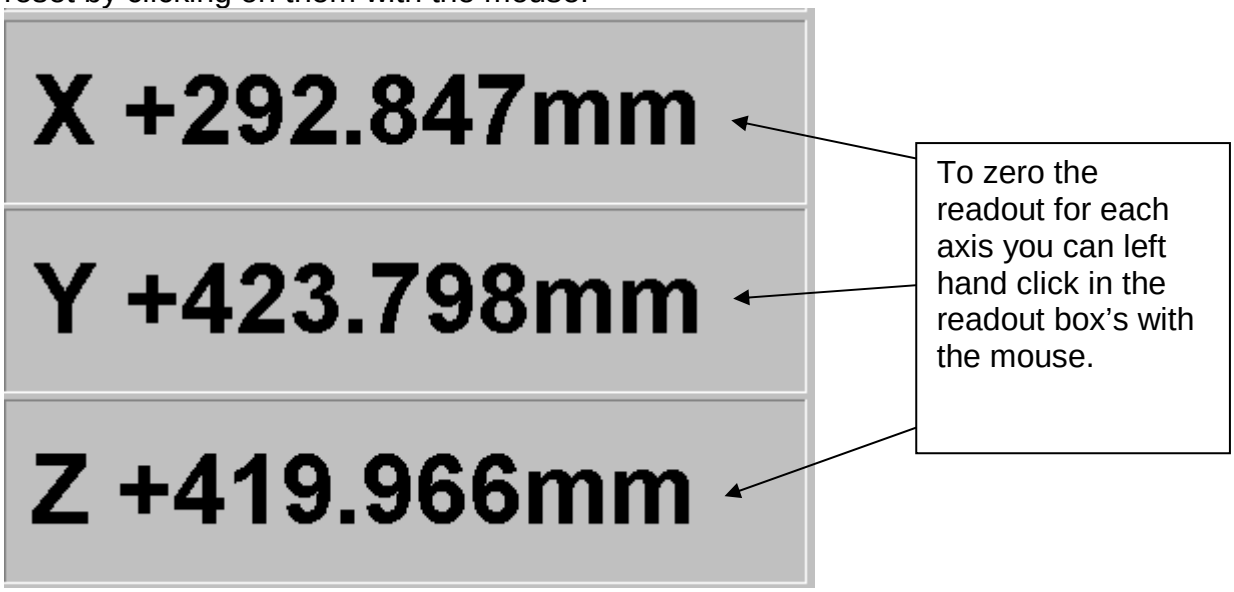
Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

www.EngineeringBooksPdf.com

22.0 QUICK MEASUREMENTS USING THE SCALE READ OUTS

It is possible to take quick linear measurements on some features by simply touching the probe on two different positions on the feature and taking the measurement from the scale readouts on the screen. These measurements however, should only be regarded as quick checks, as they cannot be saved or printed, and will only take into account single measurement points, with no stylus ball compensation.

This type of measurement is aided by the fact that the scale readouts can be reset by clicking on them with the mouse.



Hence, if you wish to make a quick check on the depth of a hole (in the Z axis), reference the plane at the top of the hole. Now gently bring the probe down until it is triggered on the surface at the top of the hole. Now with the probe still in contact with the surface, click on the Z axis readout, and the number displayed will be reset to zero.

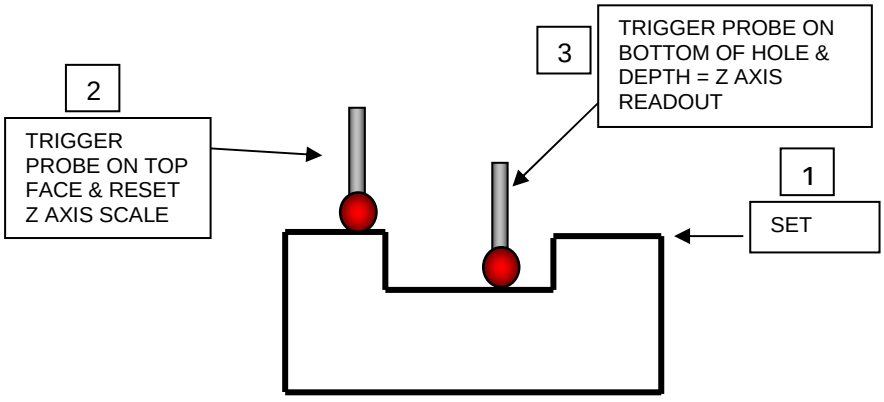


Fig. 22.0-1 A quick check on the depth of a hole

Now bring the probe down to trigger at the bottom of the hole. With the probe still in contact with the bottom of the hole, the reading on the Z axis readout will be a single point reading for its depth.

www.EngineeringBooksPdf.com

Note - This method of measurement is prone to inaccuracy if there is any run out between the top and bottom faces of the hole, as we are measuring its depth at one position only. If a more accurate method is required, or a method which can be saved or printed, then the bottom of the hole should be measured as a plane, and the distance between the top and bottom planes measured as the hole depth. In this way any run out between the two faces will be noted and averaged.

The reason that using this quick method gives the correct depth for the hole, even without any stylus compensation, is that the same side of the ball was used to take both point measurements. In actual fact the scale is reset to the centre of the stylus ball, and so both points were offset from each surface by the radius of the ball, which in this case cancels out to give the correct depth.

If however, you wanted to measure the length of a block, in the X axis say, then measure a face on the block which will define the X axis alignment, and set it as a reference. Now, in the same way as for the hole, we can touch the probe on one end of the block.

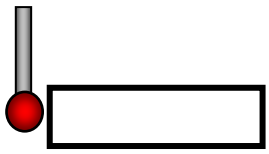


Fig.22.0-2 Touch the probe on one side of the block

With the probe in contact with the end of the block, and triggered, click on the X axis readout to zero it. Now bring the probe across and touch it on the other end of the block.

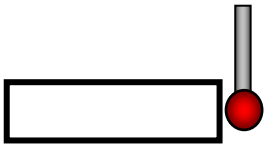


Fig.22.0-3 Then touch the probe on the other side of the block

Now in this case the reading on the X axis readout will be the length of the block plus the diameter of the stylus ball, as in this case opposite sides of the ball were used to take the measurement points.

Note that if you used this technique for measuring the separation of the internal faces of a groove, for instance, the readout reading would give you the desired separation minus the stylus ball diameter.

www.EngineeringBooksPdf.com

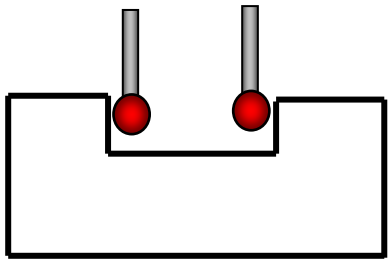


Fig.22.0-4 Measuring internal faces using the scale readouts

Remember these measurements give you the distance from ball centre to ball centre.

www.EngineeringBooksPdf.com

23.0 SAVING MEASUREMENTS

When you have completed an inspection you can save the measurements to disc either by clicking on the ‘Save File’ button or the ‘Save As’ button.

23.1 Save As

Fig. 23.1-1 The Save As button

The Save As button should be used when you wish to save the inspection to disc, allocating it a new file name. This could be simply because you are saving the inspection for the first time. When clicking on the Save As button the following window will appear.

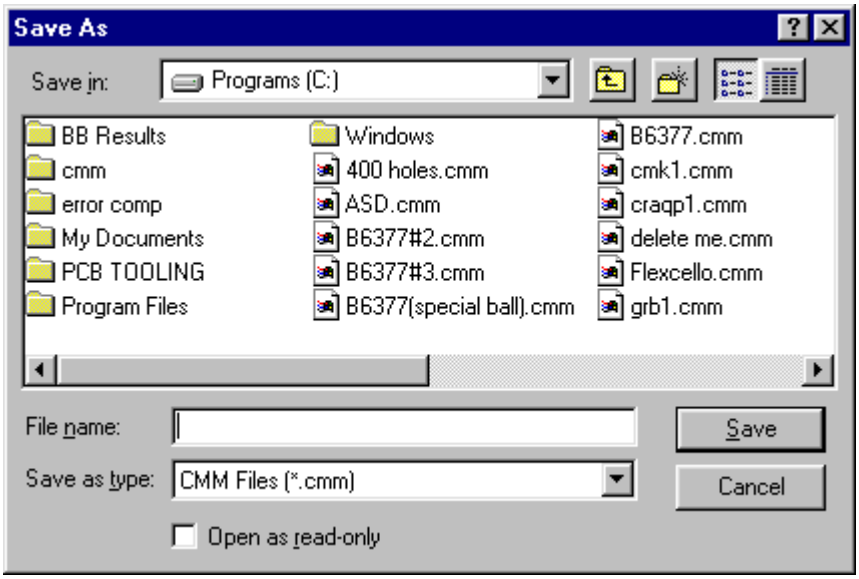


Fig. 23.1-2 The Save As window

The ‘Save In’ box will default to the file path specified in the Results Folder in the software set up, see Chapter 24 – Software Set Up. This can still be altered at this point, if desired. The ‘Save As Type’ box will default to CMM Files (*.cmm), and the cursor will be flashing in the ‘File Name’ box, waiting for the user to type in the desired file name. As the software runs under Windows 95 this can be long file name. When you have typed in the name, click on ‘OK’.

Should you type in an exact replica of an existing file name, then the following warning will appear on the screen.

www.EngineeringBooksPdf.com

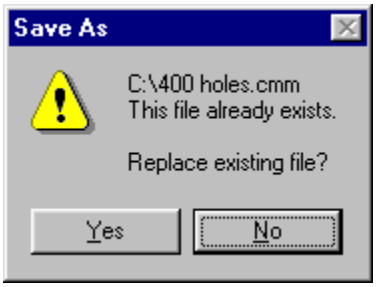


Fig. 23.1-3 The Warning Prompt that a file of the same name already exists

You can either overwrite the existing file by clicking on ‘Yes’, or if you click on ‘No’, the window will return inviting you to type in a different file name.

23.2 Save File

Fig. 23.2-1 The Save File button

The Save File button should be used when the inspection has been previously saved, and you are happy with the file name allocated to the inspection.

Note however, that any changes that have been made to the inspection since it was last saved will now overwrite the previously saved inspection, which will be lost. If you wish to keep both the changes and the previously saved inspection then you will have to save the changes under a new file name by using the ‘Save As’ button.

If you click on the Save File button and the inspection has not been previously saved, then you will be prompted to enter a file name, as for the Save As button.

www.EngineeringBooksPdf.com

24.0 STARTING A NEW INSPECTION

When the CMM is switched on at the start of the day, and then referenced (see chapter 2.0 – Referencing the Machine) the software will automatically be ready to accept a new inspection.

If you need to start a new inspection at any other time, however, you will need to click on the 'New Inspection' button.

Fig. 24.0-1 The New Inspection button

If the previous inspection has already been saved as the last action, the graphics screens will go blank, and the software will be ready to start a new inspection.

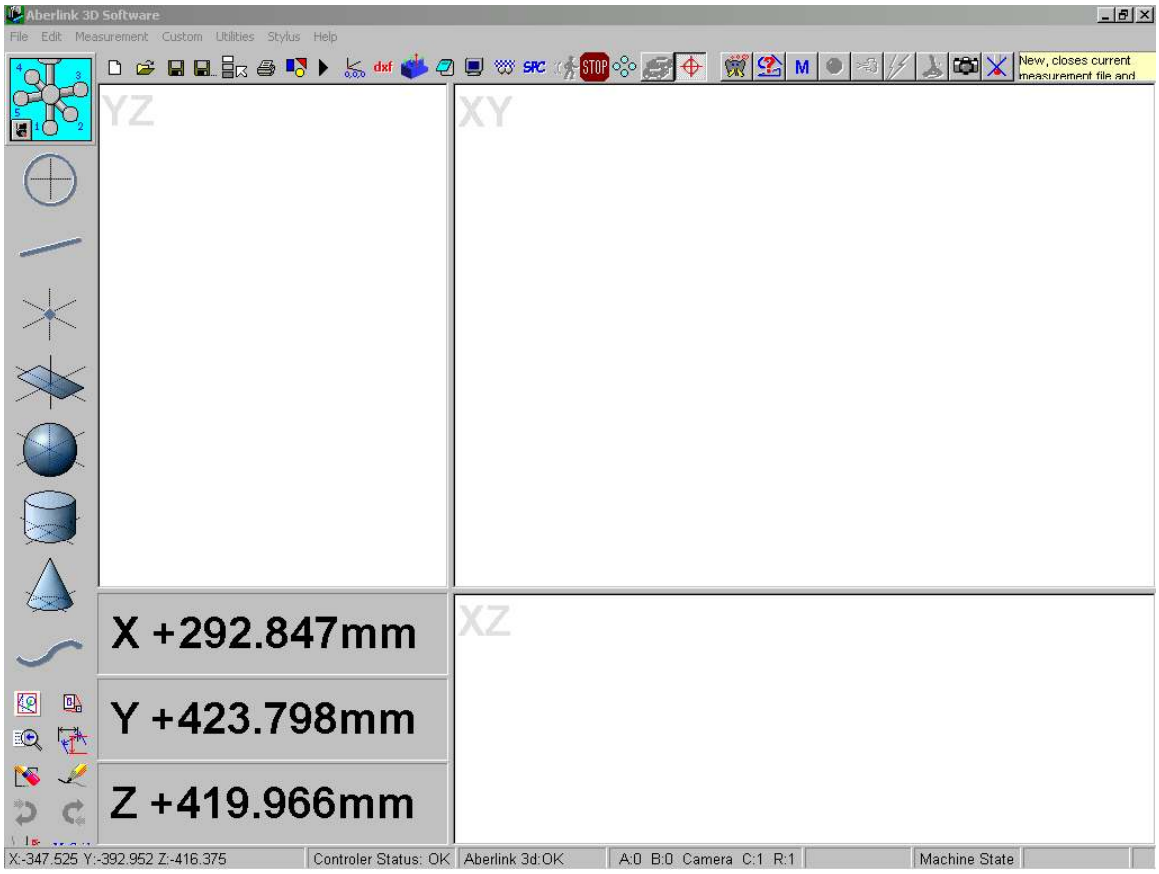


Fig. 24.0-2 The Main Screen ready for a new inspection

If however, something has been modified in the last inspection, subsequent to the last time that the inspection was saved, then the following prompt will appear.

www.EngineeringBooksPdf.com

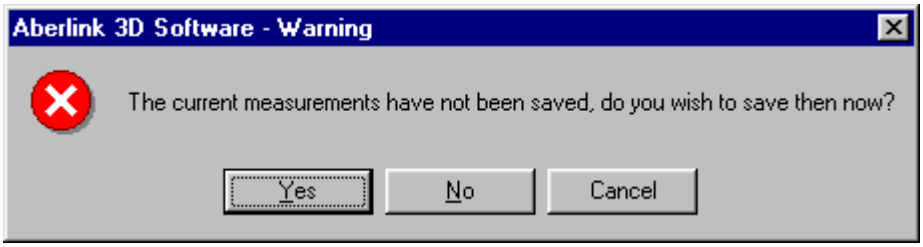


Fig. 24.0-3 The Warning Prompt that the current measurements have not been saved

You will now have three options. If you click on ‘Yes’ you will save the previous inspection, as though you had clicked on the Save File or Save As buttons, and a fresh screen will be displayed. If you click on ‘No’ the previous inspection will not be saved, but a fresh screen will still be displayed. Finally if you click on ‘Cancel’ then the screen will return to its previous state, and the new inspection button will be ignored.

www.EngineeringBooksPdf.com

25.0 OPENING A PREVIOSLY SAVED INSPECTION

If you wish to open an inspection that has previously been saved, click on the 'Open File' button.

Fig. 25.0-1 The Open File button

When clicking on the Open File button the following window will appear.

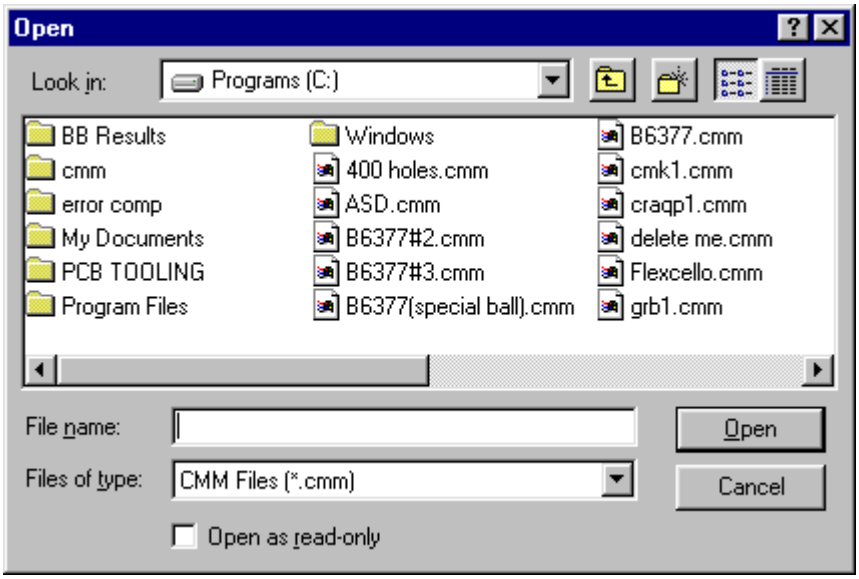


Fig. 25.0-2 The Open File window

The Look In' box will default to the file path specified in the Results Folder in the software set up, see Chapter 26 – Software Set Up. This can still be altered as necessary. The 'Files of Type' box will default to CMM Files (*.cmm), and the cursor will either be flashing in the 'File Name' box, waiting for the user to type in the desired file name, or the last opened file name will appear, but highlighted in dark blue ready to be overwritten if necessary.

Now to open the saved inspection either type in the file name required, or alternatively click on the file name displayed in the list above. When you have selected the file name, click on 'OK'.

www.EngineeringBooksPdf.com

26.0 PRINTING RESULTS

Introduction

Whenever you wish to produce a hard copy of inspection results the print function is used.

Fig. 26.0-1 The Print button

This will bring up the following window:

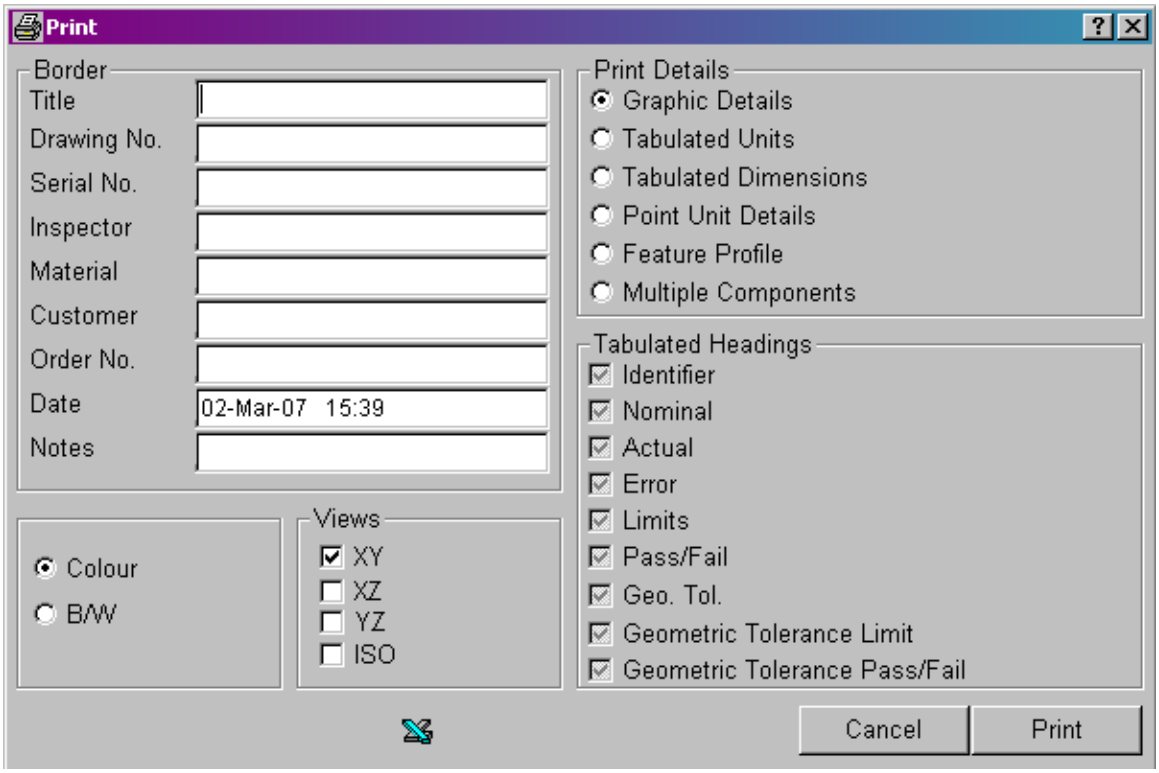


Fig. 26.0-2 The Print window

All inspection reports will be printed with a border around them, which can contain information about the part being inspected. This information can be entered in the border details frame. It is not essential to enter this information, and if not entered, the border will remain blank except for the date, which is entered by default. Your company name and address, as entered in the Software Set Up (see chapter *)* will also be printed on each page.

There are six different print routines that you can select.

www.EngineeringBooksPdf.com

- **Graphic Details,** prints results in the same form as the graphic display, i.e. an XY, XZ or YZ view of the component and dimensions.

- **Tabulated Units,** prints the details of the each feature, similar to the information in the measure window of that feature.

- **Tabulated Dimensions,** prints all dimensions that are added to report from the current or previous inspections.

- **Point Positions,** prints the co-ordinates of points and the co-ordinates and size of circles.

- **Feature Profile,** prints the graphical representation of how well the individual points taken fit the theoretical shape i.e. the profile of a surface or the roundness of a circle.

- **Multiple Components,** by printing the dimensions in a line format you can get multiple parts on one sheet of paper.

You select the type of report that you require by clicking on the relevant option button.

Printouts can be in either black and white or colour. Again to select which, you simply click on the relevant option button. If you have a single cartridge printer then it is normally more convenient to leave the black cartridge in the printer and print in black and white. However if you do have a duel cartridge printer then the colour can help highlight dimensions which are out of tolerance.

Notes (for single cartridge printers)

- If you have a black cartridge in the printer and print in colour then the printer will dither the output to produce a grey scale; this can result in very faint lines.

- If you have a colour cartridge in the printer and print only black then the printer will use all three primary colours to produce black: this can be quite expensive in ink cartridges.

Before committing to printing a report a preview of the output can be seen by first clicking on the print preview button. Only the first three sheets are shown in the print preview, and if the text or result becomes too long for the resolution of the screen then it will be replaced by a box containing a diagonal cross.

26.1 Graphic Details

This is the most commonly used print routine as it closely mimics the output that the software produces during use. When this option is selected the user

www.EngineeringBooksPdf.com

can then select one or more of the three possible views XY, XZ and YZ for printing. If more than one view is selected the individual views are printed on separated sheets. The extent of the area that is printed is based on the current software view, but as the aspect ratio of the page and screen are different it is advisable to click print preview before printing. This will confirm exactly what will be printed, especially useful if key dimensions are close to the edge of the view. If all necessary dimensions are not on the print preview then re-zoom the view in the software and recheck the print preview.

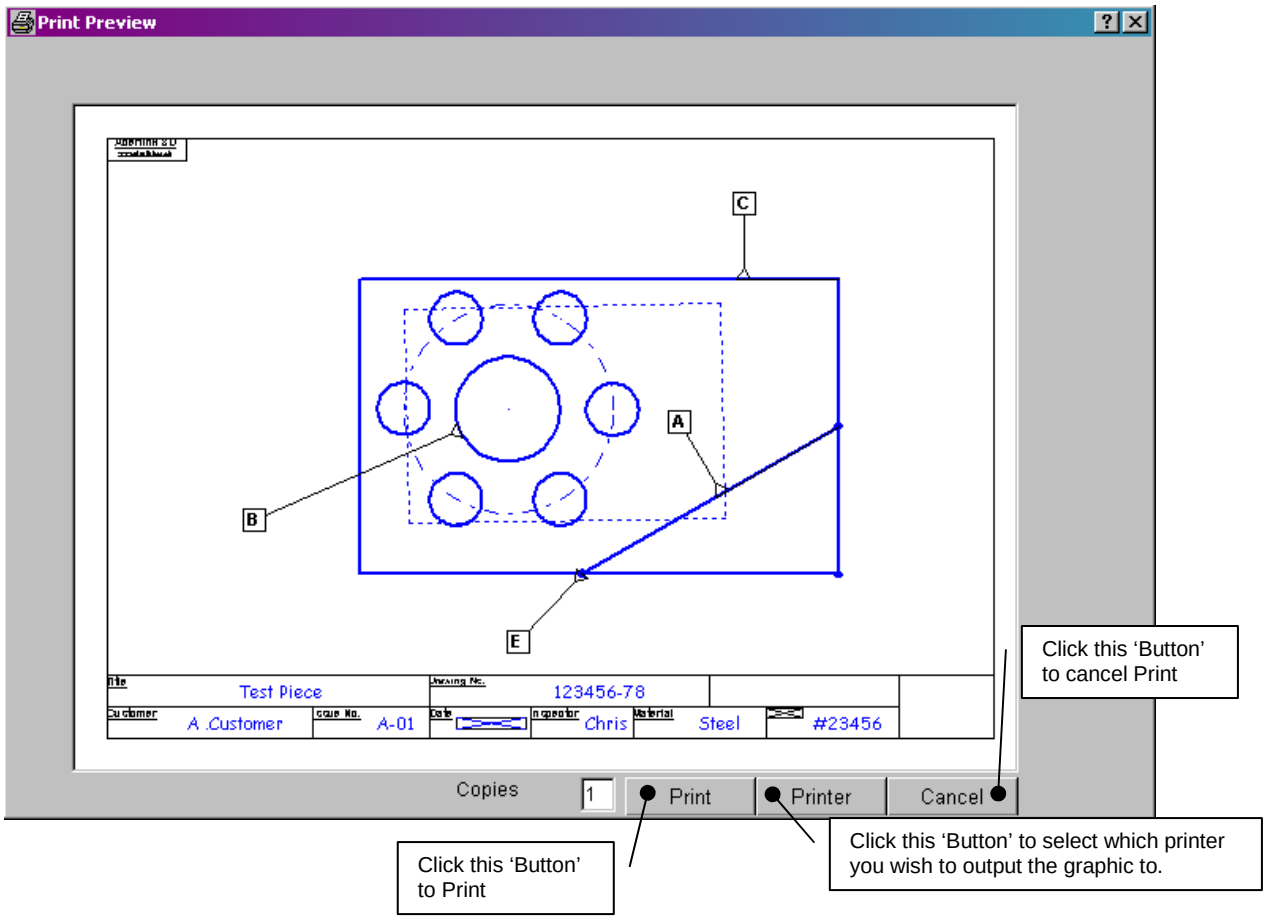
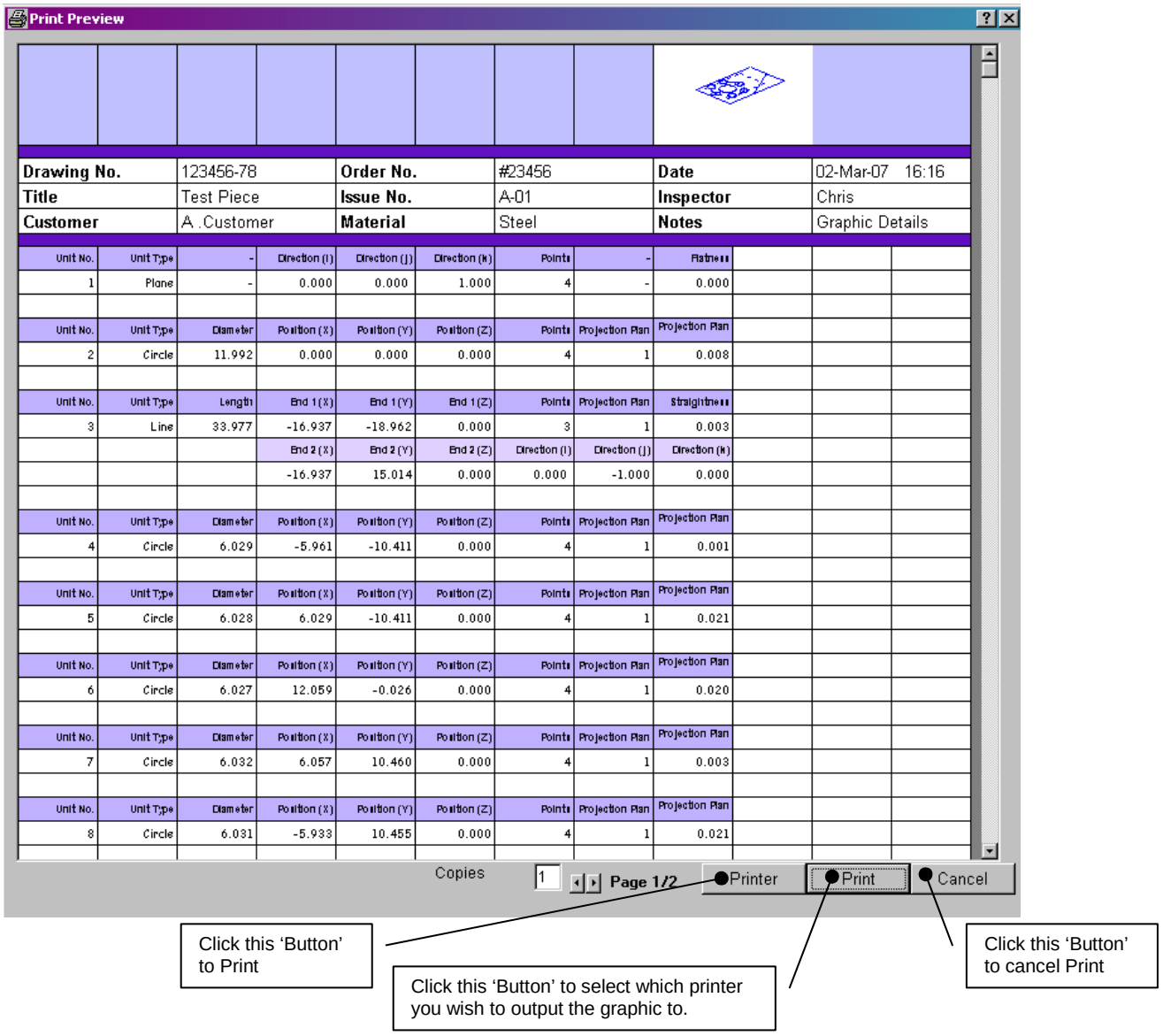


Fig. 26.1-1 A Print Preview of a Graphic Details print

26.2 Tabulated Units

This routine will print the properties of the units that have been inspected e.g. for a circle unit its centre co-ordinates and diameter will be printed or for a line unit the start and end co-ordinates will be printed, as well as its directional vector. In order for these co-ordinates to have any real meaning the part origin must have been defined, by referencing the appropriate features.

www.EngineeringBooksPdf.com



Unit No.	Unit Type	-	Direction (I)	Direction (J)	Direction (K)	Points	-	Flatness
1	Plane	-	0.000	0.000	1.000	4	-	0.000

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
2	Circle	11.992	0.000	0.000	0.000	4	1	0.008

Unit No.	Unit Type	Length	End 1 (X)	End 1 (Y)	End 1 (Z)	Points	Projection Plan	Straightness
3	Line	33.977	-16.937	-18.962	0.000	3	1	0.003
			End 2 (X)	End 2 (Y)	End 2 (Z)	Direction (I)	Direction (J)	Direction (K)
			-16.937	15.014	0.000	0.000	-1.000	0.000

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
4	Circle	6.029	-5.961	-10.411	0.000	4	1	0.001

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
5	Circle	6.028	6.029	-10.411	0.000	4	1	0.021

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
6	Circle	6.027	12.059	-0.026	0.000	4	1	0.020

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
7	Circle	6.032	6.057	10.460	0.000	4	1	0.003

Unit No.	Unit Type	Diameter	Position (X)	Position (Y)	Position (Z)	Points	Projection Plan	Projection Plan
8	Circle	6.031	-5.933	10.455	0.000	4	1	0.021

Fig. 25.2-1 A Print Preview of a Tabulated Units print

26.3 Tabulated Dimensions

The dimensions that have been added to a part inspection can be printed in tabulated form. This is often more useful than printing Tabulated Units as these dimensions should be the same as on the original component drawing.

When printing Tabulated Dimension reports, it is possible to select what information is printed in the report using the check boxes in the Print window. Note that if you wish to print nominal values, then the software will simply use the measured values rounded to the precision set in the Software Set Up, and default tolerances will be plus or minus half of this rounding value, unless you

www.EngineeringBooksPdf.com

have typed in a value for the nominal and tolerance using the Dimension Detail box (see chapter 16.8 – Setting Nominal Values and Tolerances for Dimensions).

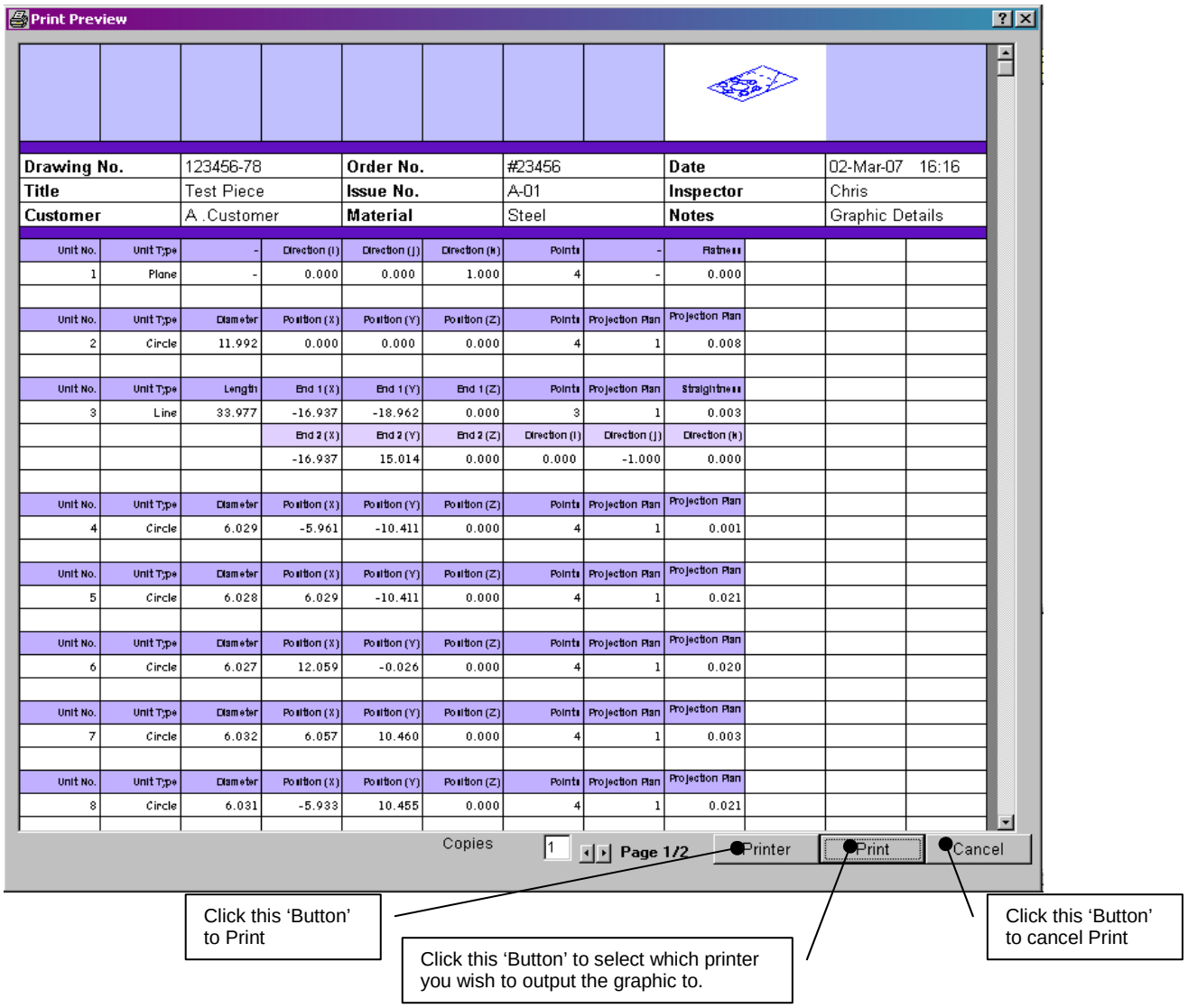


Fig. 26.3-1 A Print Preview of a Tabulated Dimensions print

It is possible to add the dimensions from more than one inspection to a Tabulated Dimensions report, which is why the 'Clear Report' and 'Add to Report' buttons appear on the Print window. Every time you click on the 'Add to Report' button, the dimensions that are displayed on the main screen will be added to the report. This will continue until the 'Clear Report' button is clicked.

www.EngineeringBooksPdf.com

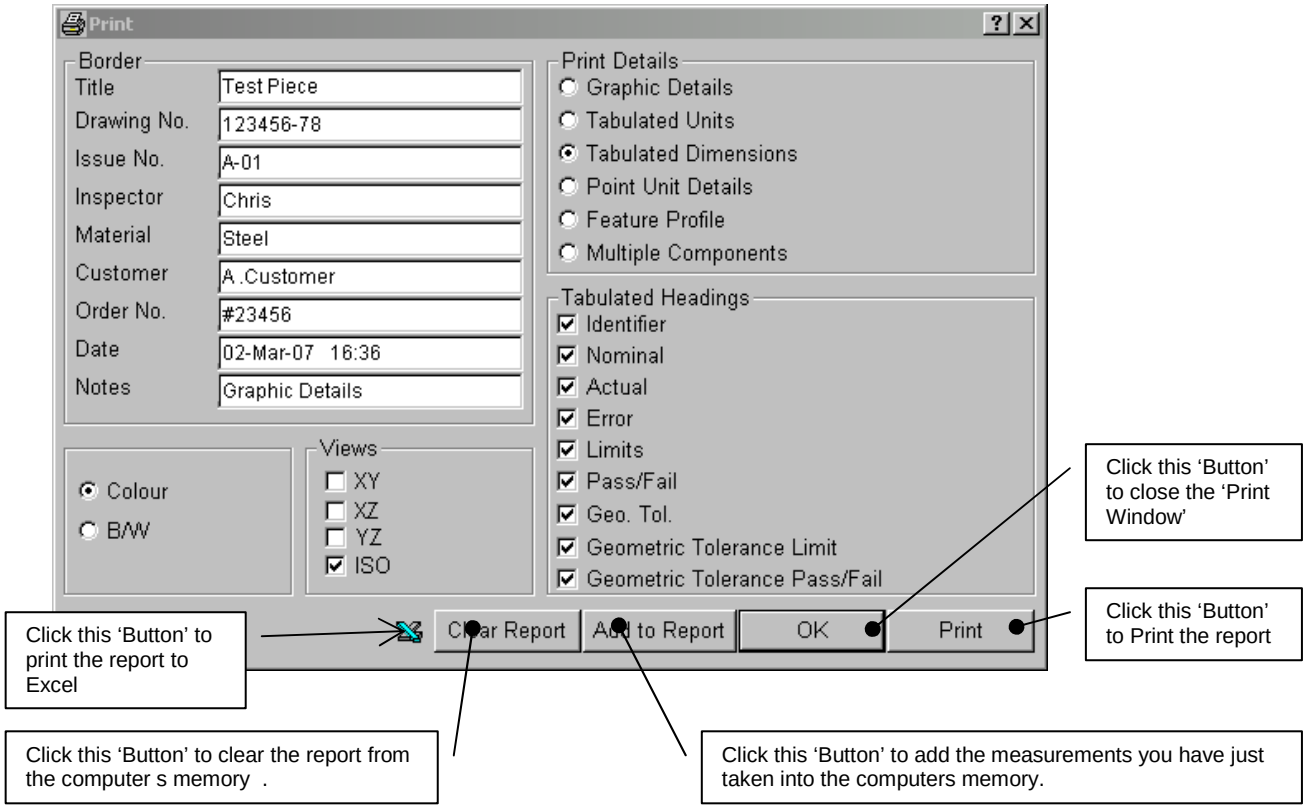


Fig. 26.0-2 The Print window showing tabulated dimension

26.4 Point Positions

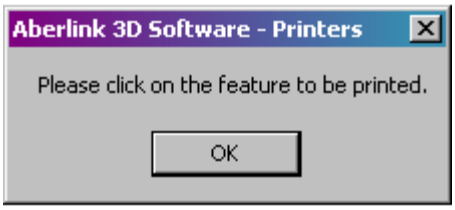
This print routine is similar to the Tabulated Units routine; the main difference is that only the X,Y position and diameter of circles and points are printed. This is very useful if you are only interested in hole positions, say if inspecting a PCB or a drill jig. If you wish to print the positional error then the nominal position of each of the holes must be checked; this is done by clicking on the down chevrons in the measurement window (see chapter 8.1.2 – Extra information about measured circles)

Note that once again, the amount of information printed can be selected using the check boxes in the Print window.

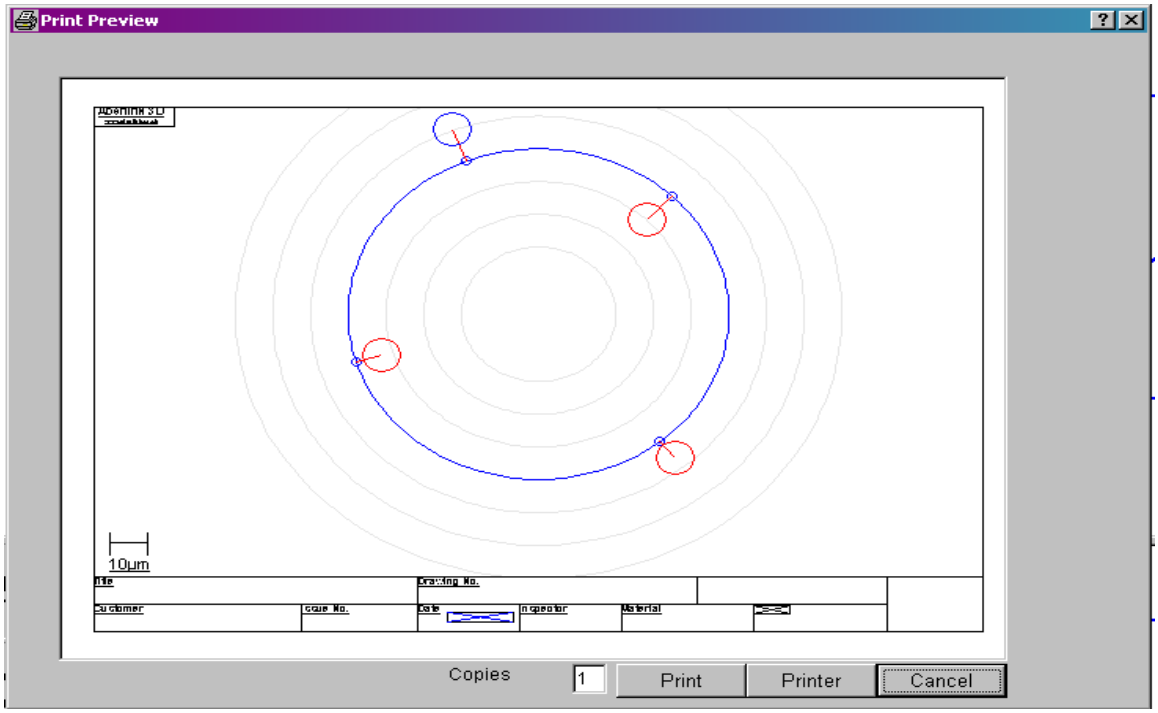
26.5 Feature Profile

As a feature is inspected, the fit of the individual points to the theoretical shape is shown in the picture box on the right side of the measurement window. If enough points are taken then a useful representation of the true surface is built up. This picture can be printed using the feature profile routine. When you click on Print or Print Preview buttons, then the software asks you to click on the feature you wish to print.

www.EngineeringBooksPdf.com



When you click on OK, the feature profile will be displayed as follows:



26.6 Multiple Components

If when measuring a batch of components using the 'Play' function with 'SPC All Components' ticked on, the Aberlink 3D software will collect all the measured data for the entire batch. These values can be printed out in a single report using this option:

www.EngineeringBooksPdf.com

Print Preview

ABERLINK LTD.

Avening Mill
Avening
Glos. GL8 8LU UK

Drawing No.	1252454	Description		Date	05-Mar-07 9:26
Title	Small Block	Issue No.	A	Inspector	Chris
Customer		Serial No.		Notes	

Item No.	34mm Between	55mm Between	30° From Line to	12mm Circle	15mm Between	17mm Between
1	34.062	54.944	30°06'24"	11.964	~~18.997~~	~~16.952~~
2	34.060	54.945	30°07'20"	11.965	~~18.995~~	~~16.956~~
3	34.060	54.944	30°07'20"	11.967	~~18.995~~	~~16.954~~
4	34.061	54.944	30°07'35"	11.966	~~18.995~~	~~16.955~~
5	34.061	54.943	30°07'37"	11.966	~~18.996~~	~~16.954~~
6	34.061	54.944	30°07'32"	11.966	~~18.996~~	~~16.954~~
7	34.060	54.944	30°07'21"	11.966	~~18.994~~	~~16.954~~
8	34.059	54.944	30°07'29"	11.966	~~18.994~~	~~16.954~~
9	34.059	54.942	30°07'24"	11.967	~~18.995~~	~~16.953~~
10	34.060	54.943	30°07'20"	11.966	~~18.996~~	~~16.954~~

Copies 1 Page 1/1 Printer Print Cancel

Note:- that any dimensions out of tolerance will be shown in **red.**

www.EngineeringBooksPdf.com

27.0 THE PLAY FUNCTION – running a program automatically

A program can be created either in 'Teach and Repeat' mode by actually measuring a component with the CMM, or by programming features 'Off Line' typing in the size and position of features, or by a combination of both methods. This was previously discussed in chapter 7 – Creating a CNC Program – a general introduction.

When you are happy with the program and ready to run it automatically click on the Play button:

It is important if your program is to run successfully that the position and alignment of the component has been properly defined by setting features as references, see chapter 6 – Defining the Alignment of a Component.

The golden rules for choosing reference features are as follows:

1. References must be selected that fully define all 6 degrees of freedom for a component when it is placed on the table. Usually this will be done using 1 plane, 1 line and 1 circle or point. However, if these features do not exist on your component other alternatives are available.

2. Measure the reference features as close to the start of your program as possible. When running a program automatically the exact position and orientation of a component will not be defined until all the reference features have been measured.

3. Choose reference features that are easy to measure eg. Large planes, edges or holes, rather than important features that are perhaps defined as datums on the drawing. Remember setting a feature as a datum is totally different to setting it as a reference – reference features are simply used to define the position and alignment of the component.

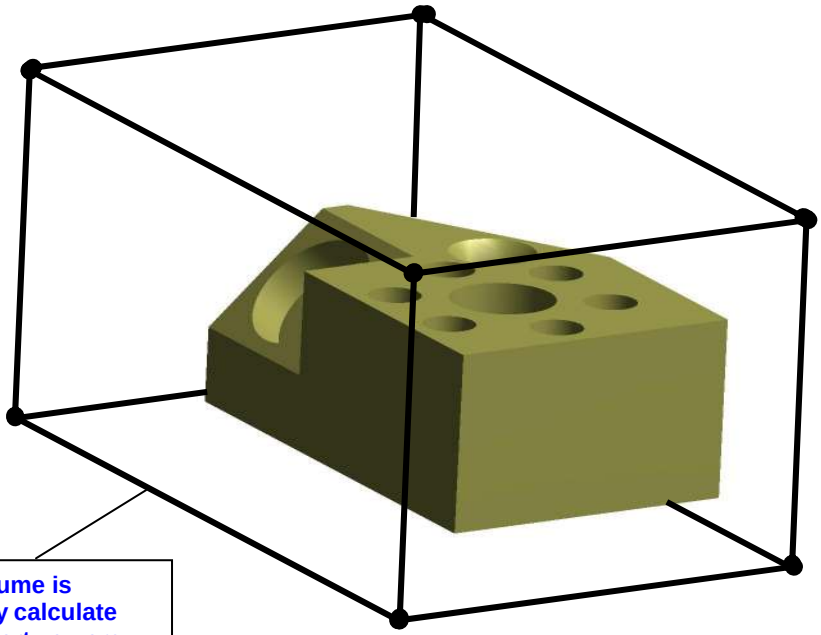
4. Once you click on the Play button it is not possible to modify or change your reference features. So once you have measured your reference features at the start of the program do not touch them again

27.1 The Safe Volume

When you are running a program for the first time, having clicked on the Play button there are 3 things that the software must do before the program can run. The first thing is to define a 'Safe Volume' around the component through which the probe must not travel when moving between features or when indexing. When you click on the Play button the following prompt will appear:

www.EngineeringBooksPdf.com

In almost all cases it will be acceptable to automatically calculate the safe volume by clicking on 'Yes'. The default size of the safe volume is defined in the Machine Set Up – see chapter 26. Now the program will automatically move to the edge of the safe volume between measuring units to avoid crashing into the component.



The Safe volume is automatically calculate around the part you are measuring

Note that the safe volume can only be created around features that have been measured. If your component has protrusions that are not measured as part of the program you may need to adjust the safe volume to account for them.

The safe volume is always infinite in a negative Z direction, so the probe can never go underneath the part being measured.

The values of the safe volume are shown in the workpiece co-ordinate window – see section 27.5 of this chapter.

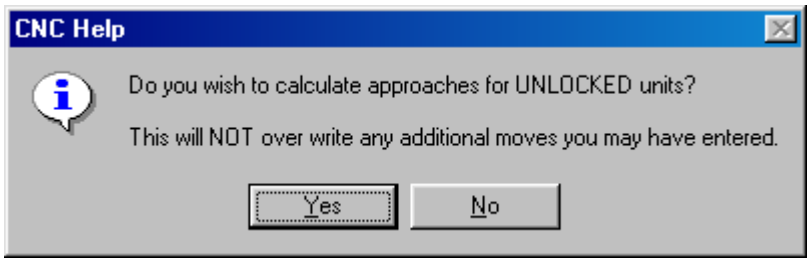
27.2 The Approaches

The position that the probe goes to prior to measuring a unit is known as the 'Plunge' position. This position we be directly above the feature being measured on the edge of the safe volume. When it has finished measuring the feature it will also return to the edge of the safe volume. This position is

www.EngineeringBooksPdf.com

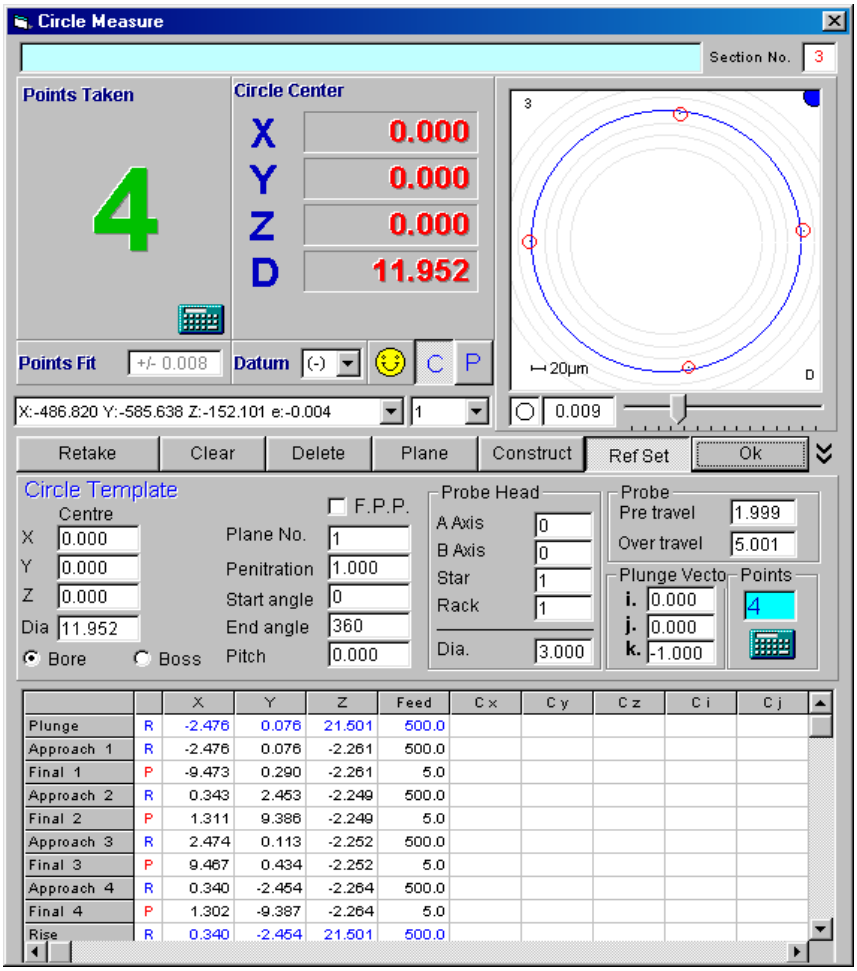
known as the ‘Rise’ position. Collectively the Plunge and the Rise are known as the Approaches for the measurement unit.

Having calculated the Safe Volume as described in section 27.1 above, the software must now calculate the Plunge and Rise values for all measurement units. The following prompt will appear:



Again it will almost always be correct to let the software calculate the approaches by clicking on ‘Yes’.

The Plunge and Rise values are stored in the probe moves grid for each measurement unit. The Plunge move will be the very first line, and the Rise will be the very last:

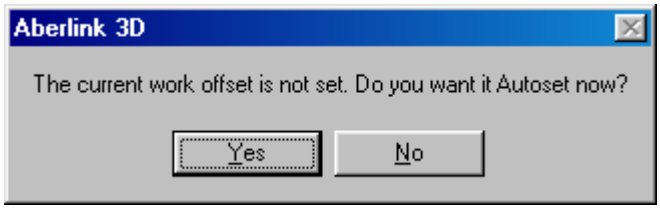


www.EngineeringBooksPdf.com

It is possible to modify these values by simply overwriting them in the grid, but do not then re-calculate the template, simply OK the unit. Similarly if you answer 'Yes' again to the original prompt these values will be automatically calculated again and overwrite your modifications.

27.3 The Workpiece Co-ordinate (or WPC)

Having automatically calculated the Safe Volume and Approaches for the program, the third piece of information required to automatically run the program is the workpiece co-ordinate, (sometimes called the work offset), and the following prompt will appear on the screen:



The reference features that you will have measured will have defined a 0,0,0 position on the component being measured. (This will be identified by a small datum symbol on the Main Screen). The workpiece co-ordinate defines where this position on the component is within the machines measurement volume – ie. where the component is on the table.

If the component has not been moved since the program was created (and the reference features measured), or it has been put back on the table in the same position (perhaps using a fixture to locate it) then the workpiece co-ordinate (or WPC) can be automatically calculated, so you can simply click on 'Yes'

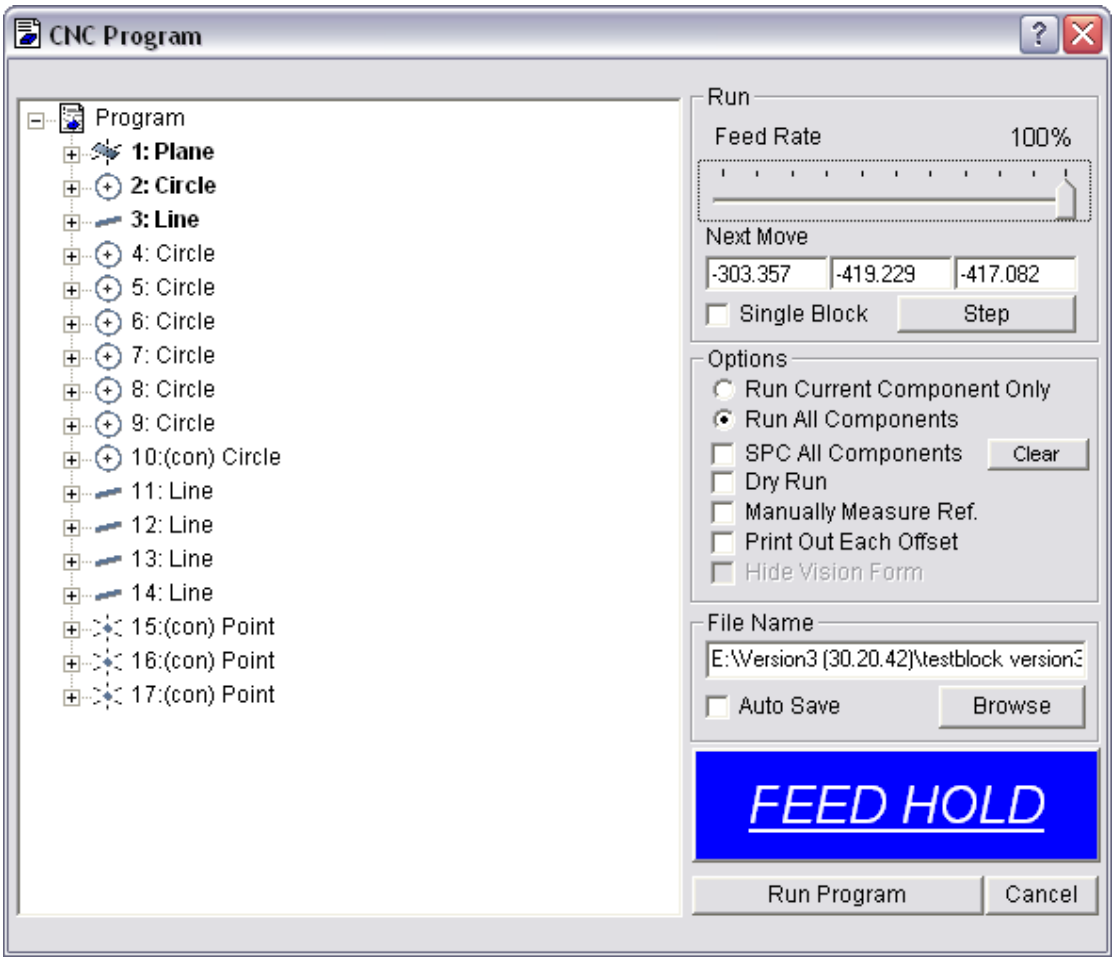
The X,Y and Z values for the WPC are stored along with the safe volume in the workpiece co-ordinate window – see section 27.5 later in this chapter.

If you have moved the component since the program was created you must click 'No' and define the WPC as discussed in section 27.5.

www.EngineeringBooksPdf.com

27.4 The CNC Program Window

When you have defined the Safe Volume, the Approaches and the WPC the CNC Program window will appear as follows:

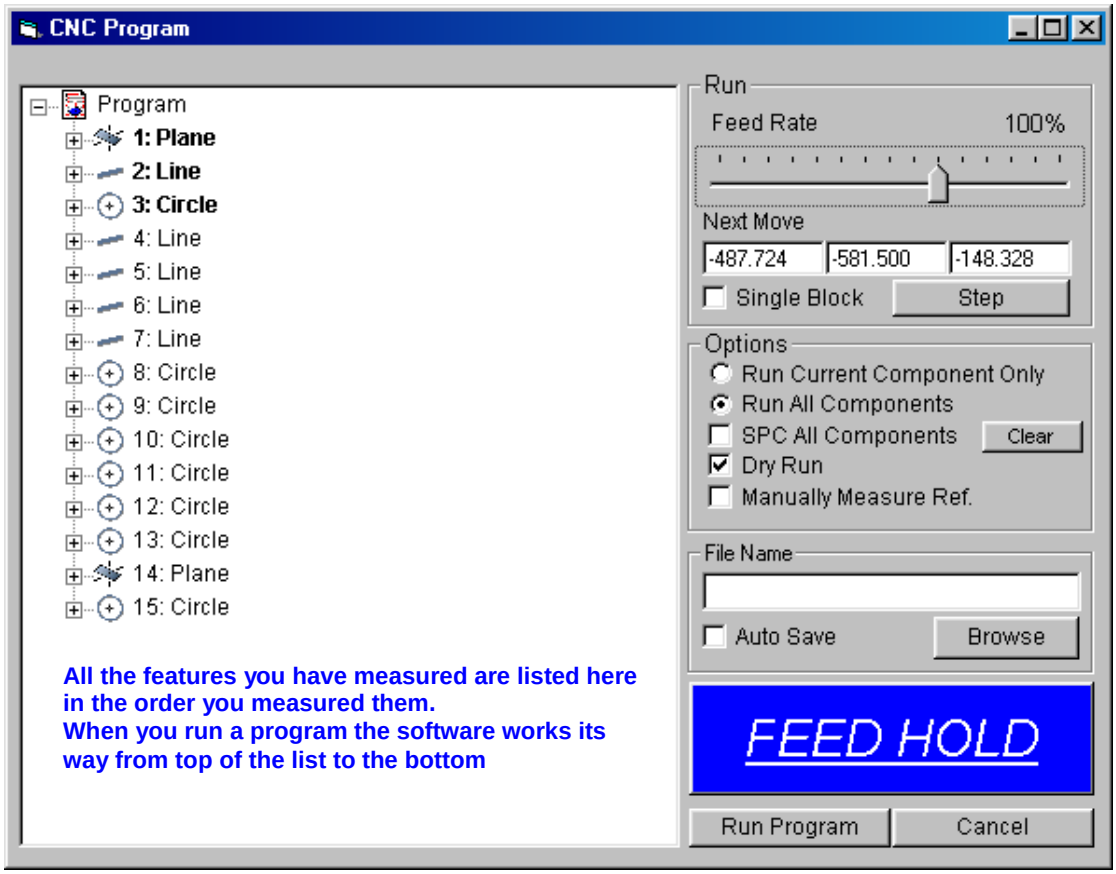


27.4.1 The Program Tree

All the measurement units are shown in the program tree on the left hand side of the window. The reference features will be shown in bold.

More information about the units can be obtained by clicking on the plus sign adjacent to it. It is possible to look at the template for the unit and even the probe moves that will be made, although it is not possible modify the moves at this window. This must be done by recalling the unit from the main screen and modifying the probe moves grid within the measurement window.

www.EngineeringBooksPdf.com



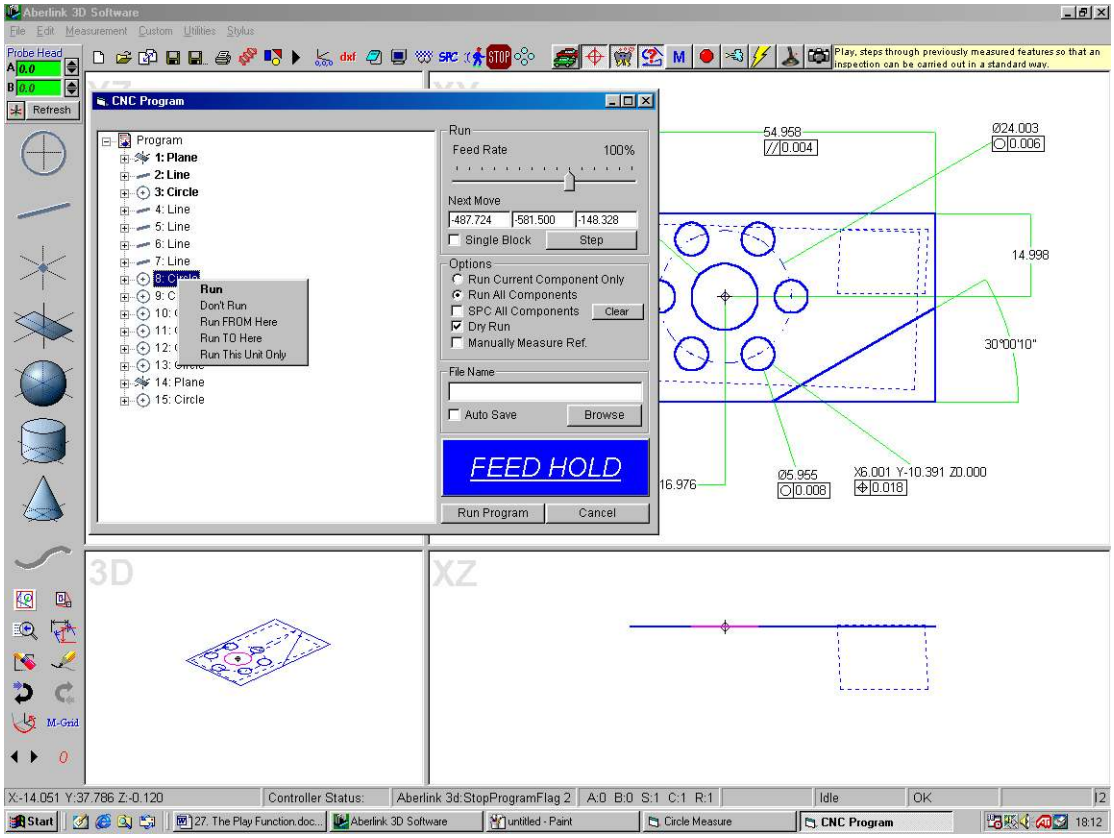
You can modify the order of the program by clicking on a unit within the tree and dragging it to a new position. Care however, must be taken not to infringe the basic rules of the Aberlink 3D measuring software such as dragging a circle or a line in front of the Plane into which they are projected, for instance.

It is also recommended not to move the position of the reference features.

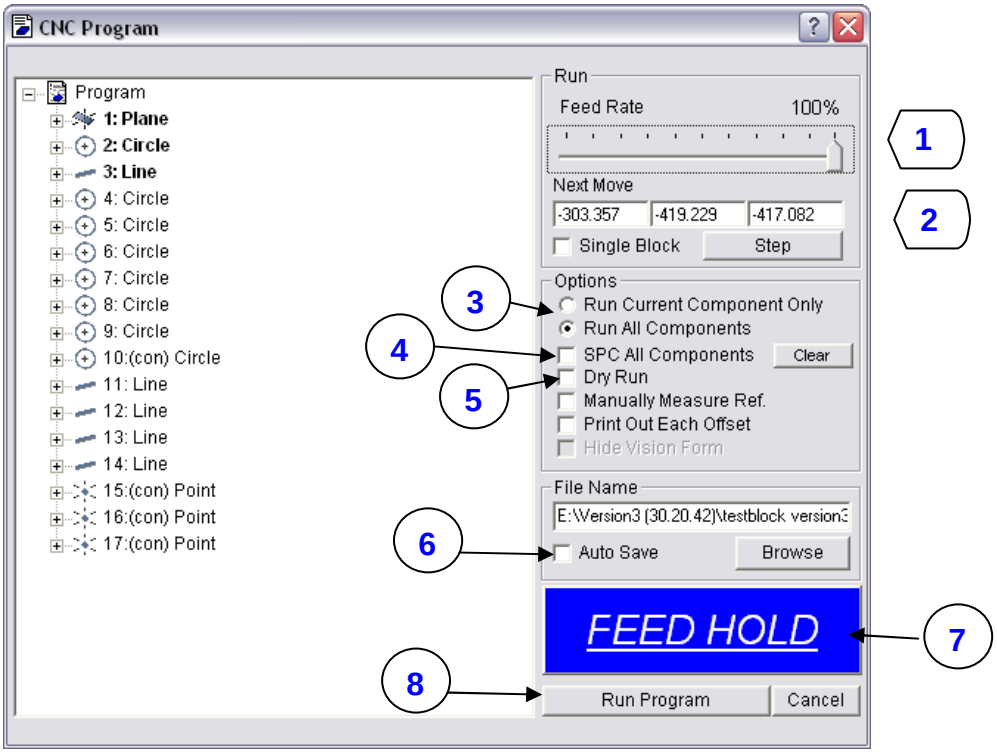
Finally you will discover that it is not possible to drag a unit to the very bottom of the tree. If you need to do this you will have to drag the unit as low down the tree as possible (to the penultimate position) and then drag the last unit above it.

Also it is possible to run only part of a program, if desired. Right clicking on a unit on the tree will bring up the following menu:

www.EngineeringBooksPdf.com



The program can be edited by either running or not running a unit, running the program up to a unit or from the unit, or even running a single unit only. You should only choose not to run the reference features if they have been measured previously and the component has not been moved, otherwise the exact position and alignment of the component will not have been defined.



www.EngineeringBooksPdf.com

1 Feed Rate

It is possible to adjust the feed rate of the probe moves using the slider bar in the top right of the CNC Program window. This may be useful to slow a program down when initially proving it out.

2 Single Blocking a Program

You can also choose to step a program through each individual move by ticking the Single Block box. When you run the program the next X,Y,Z move for the probe will appear in the white boxes and the probe will move there when you click on the Step button.

3 Run Current Component only or All Components

It is possible to run a single program on multiple components by setting the workpiece co-ordinate for each component as discussed in section 27.5 below. Under the Options in the CNC Program window it is possible to select for the program to run the current work offset only or alternatively all the work offsets that have been set.

4 Collecting SPC Batch Information

If you wish to collect SPC batch information for a batch of components tick the SPC All Components box. The information collected is discussed in chapter 34 – SPC Batch Information.

5 Dry Run

By default when you first go into the CNC Program window the Dry Run option will be ticked. If you click on the Run Program button now you will get a simulation of the probe moves with a red circle representing the stylus ball moving around the main screen where the probe will move. This may be useful for spotting programming errors.

6 Auto Save

If you are running multiple components each subsequent time that the program runs it will overwrite the previous inspection. If the SPC box is ticked all the measurement results will be saved to the batch information but individual reports for the overwritten components will not be possible unless you tick the Auto Save box.

Now each time the component is run the software will automatically save a file for it. For instance if the program has been saved under the name of ‘Test Piece’ the first time the component is run the results will be saved as Test Piece #1 and then Test Piece #2 etc. Now when the batch finishes if you wish to print an inspection report for the first component you can now open the Test Piece #1 file and create the desired report.

www.EngineeringBooksPdf.com

If the program has not been saved then you will automatically be prompted for a name when you try and run the program.

7 Feed Hold
If you 'click' on this button the CMM will pause the program if you 'click' the button again the CMM will carry no from were the program stopped.

Manually measure Ref.

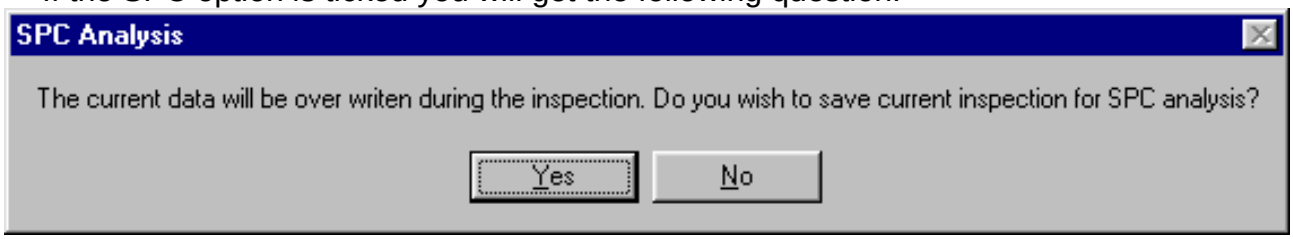
If you run a program with this box tick the features that you used to align the part (mark in **bold letters** in the program tree) turn green in the main window and must be measured manually before the program can run.

27.4.8 Run Program and Cancel (8)

If you are happy that the program is correct and that the safe volume, approaches and WPC have all been correctly set then, if you un-tick the dry run option and click on Run Program the CMM will now drive round and measure the component automatically.

Note that the position that the program is started from is the position that it will return to at the end. This position is known as the Home Position. It is advisable to start a program from a safe position above the component. If you are measuring multiple components start the program from above the first component. If the program fails to run successfully you will be asked 'Do you wish to return to the Home Position?' If you answer 'Yes' the probe will return to the home position travelling in Z first. If this path is blocked you must answer 'No'

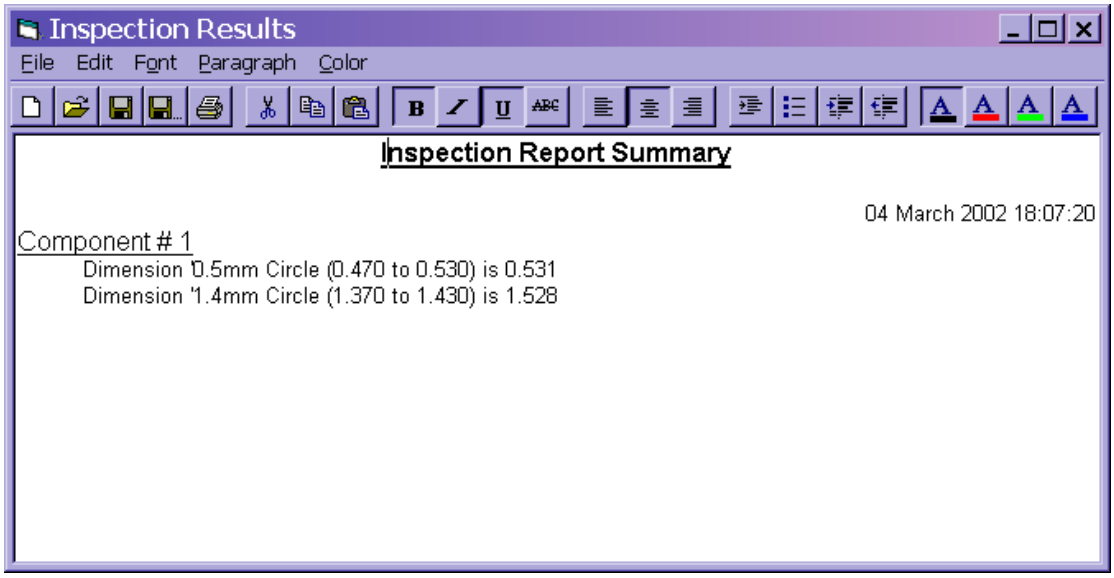
If the SPC option is ticked you will get the following question:



This is because the data on the screen is about to be overwritten and you need to decide whether you require these results to be added to the batch information.

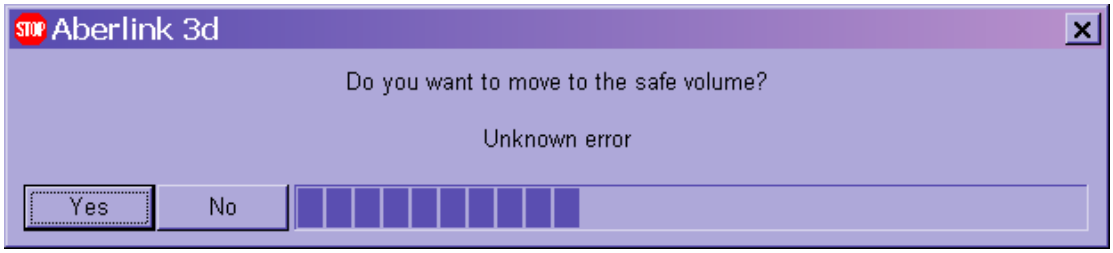
If the program runs successfully a summary of the inspection will appear on the screen listing any dimensions that are out of limits:

www.EngineeringBooksPdf.com



Cancel will close the window and the program will not run.

During the running of the program the CNC Program window is minimised to the bottom of the screen. If you wish to cancel the program while it is running click on the minimised button to bring the window back up and click on Cancel. The following prompt will appear:



If you answer ‘Yes’ to this question the probe will get away form the component moving to the edge of the safe volume.

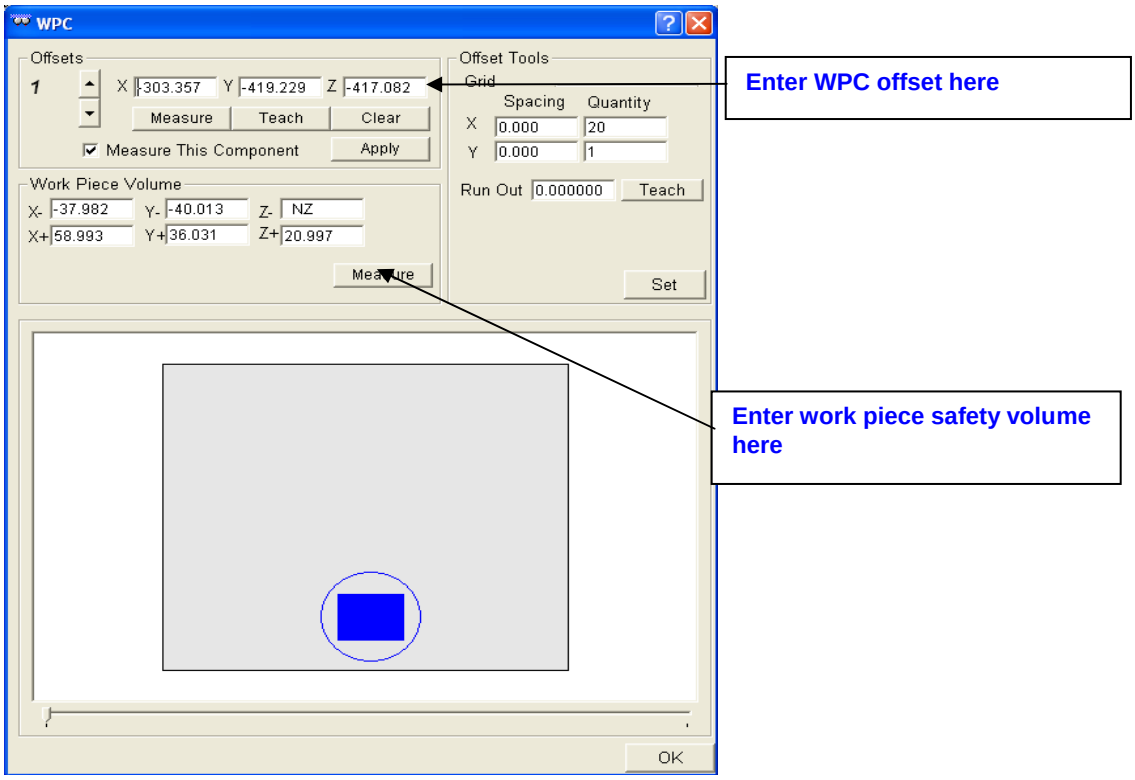
Follow the rest of the prompts until the program is finally cancelled.

www.EngineeringBooksPdf.com

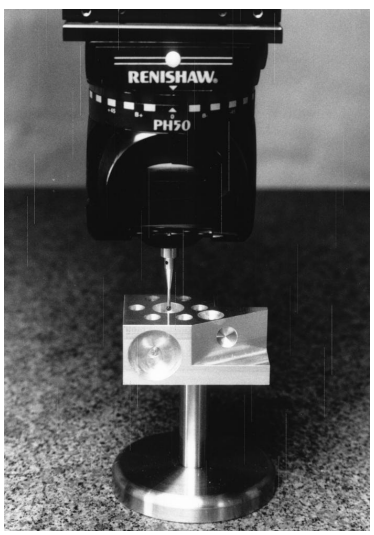
27.5 The work piece co-ordinate (WPC or Work off set)

Most of the time the work piece co-ordinate and the safe volume will be automatically set by the software. These values are stored in the workpiece co-ordinate window:

The X, Y, Z work piece co-ordinate values are shown in top left hand section of the WPC window.



If the component has been moved on the table since the programme was created then it will not be possible to automatically calculate the WPC and therefore you will have to define it as follows:



Drive the probe so that the ruby ball is physically at the position of the datum on the component. If your datum is a hole, the ruby ball should be at the centre of the hole so that the centre of the ball is at the top face of the hole:

If you have chosen a point on a plane (such as the corner of a block) so that this is not possible, drive the ruby ball so that its bottom is just over the corner of the block – we will have to adjust in Z by the radius of the ball later.

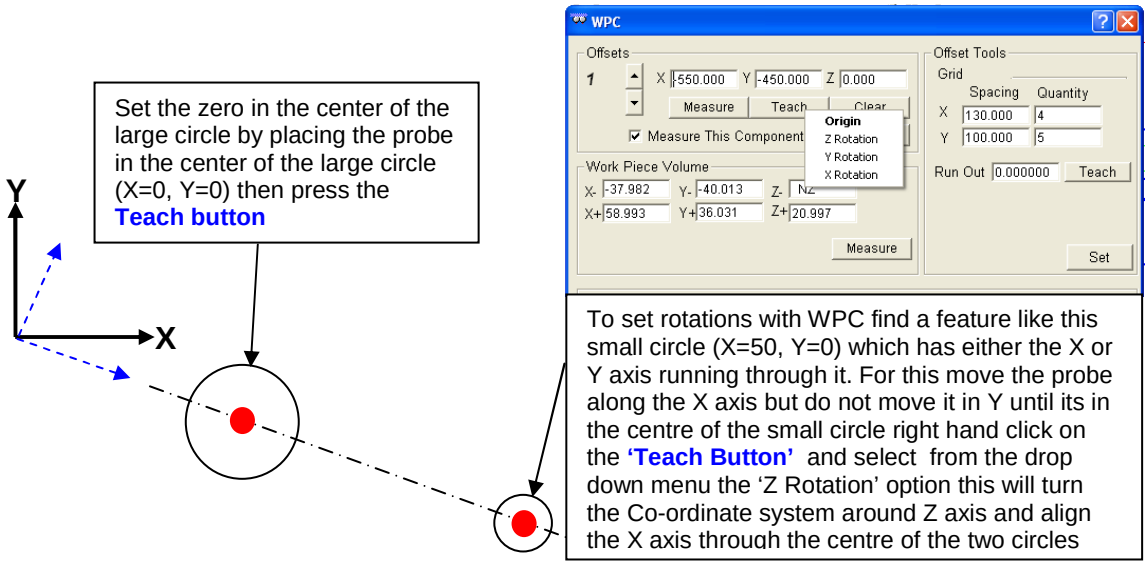
www.EngineeringBooksPdf.com

Note that you only have to set the WPC accurately enough so that the programme will find the datum features. Once these have been measured automatically in the program the WPC will be updated to an exact value.

Tick the 'Measure this Component' box and then click on the 'Teach' button. The position of the ruby ball will then automatically be entered as the WPC. If you have positioned the ball away from the datum on the component (eg above the corner of the block) then you will now have to modify the values accordingly.

The 'Measure' button will automatically calculate the WPC from the reference features that have been measured in the current inspection. In effect this button will perform the same calculation as the calculation performed when prompted to automatically calculate it when running a programme for the first time. It assumes that the component has not been moved since the last time it was measured.

27.6 Setting rotations with WPC



Set the zero in the centre of the large circle by placing the probe in the centre of the large circle (X=0, Y=0) then press the 'Teach Button' This will automatically set the position centre of the ball as the new WPC.

To set rotations using WPC window find a feature like the small circle (X=50, Y=0) which has the X or Y axis running through it. Move the probe along one of the axis (like X) but do not move it in the other (like Y) until the probe in the centre of the other feature (both feature lie along the same axis.)

If you right hand click on the 'Teach Button' you see a drop down menu with four options origin, Z rotation, Y rotation and Z Rotation select the 'Z Rotation'

www.EngineeringBooksPdf.com

option this will rotate the co-ordinate system around Z and align the X axis through the centre of the 2nd feature which lie along the same axis as the WPC.

Problems you can have with WPC

To run a program it's important to have the correct WPC (datum) entered into the offset box in the WPC window. If the wrong co-ordinates have entered the machine will not be able find the part on the table and could crash into the part or table.

Note: Sometimes when loading a programme the WPC can set itself to zero. If you tried to run the programme with the WPC at zero the machine will try to move to the reference position (zero position machine co-ordinate system) in the top right hand corner of the CMM.

If you are not using rotation option when setting a WPC you must place the parts on the table square to the machine axis system

If you have positioned the ball away from the datum on the component (e.g. above the corner of the block) then you will now have to modify the values accordingly by taking away half the diameter of the ball.

27.6.1 Setting Multiple Workpiece Co-ordinates (multiple components)

To the left of the WPC values are 2 black arrows and a number (initially 1). If you click on the upper black arrow the number will change to 2 and the offset values for X, Y and Z will be refreshed.

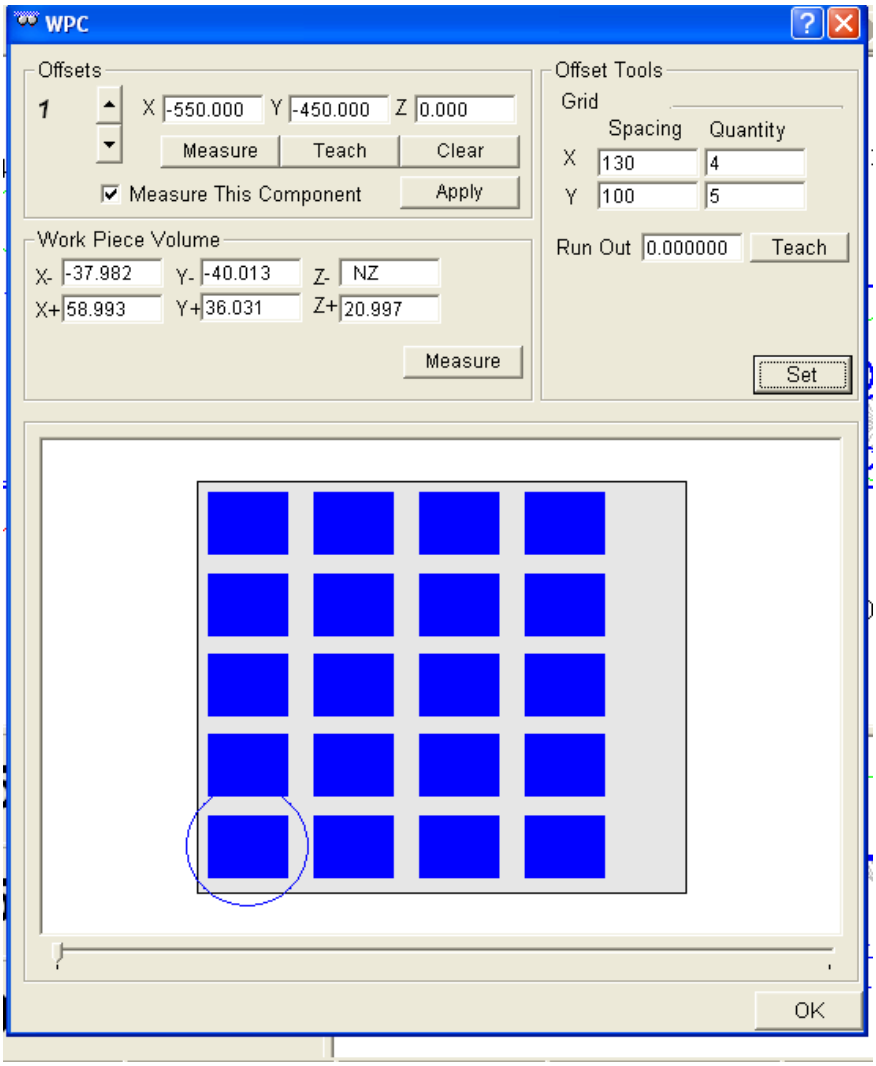
You can now set the WPC for a 2nd component in exactly the same manner as for the first component as described above. As long as you tick the 'Measure this Component' box for item 2 the program will automatically run both components when you run it. You can continue to set as many workpiece co-ordinates as you like and control which positions are run using the 'Measure this component' tick box.

27.6.2 Automatically setting work offsets for a grid of components

If you are using a fixture that holds a grid of components at a regular spacing then it is not necessary to set the WPC for each component. Simply set the WPC for position 1 and then using the 'Grid' section to define the spacing and quantity of components in both the X and Y directions.

For example, if you had a fixture that held 12 components in a 4 x 5 grid with the X spacing at 130mm and the Y spacing at 100mm, then fill in the grid details as follows:

www.EngineeringBooksPdf.com



Now when you click on the ‘Set’ button the other 19 work offsets will automatically be filled in (relative to position 1) and their ‘Measure this Component’ boxes will be ticked in the WPC window.

At the bottom of the WPC is a graphic showing the offsets as they would be spaced out on the CMM table, the current work piece offset is always shown with a circle around it.

Note that if you don’t have a full fixture of components you can control which positions are run by un-ticking the relevant ‘Measure this Component’ box.

27.6.3 The Safe Volume Values

Most of the time the software will automatically calculate the safe volume. These values are stored in the WPC window in the bottom left hand section under ‘Workpiece Volume’

www.EngineeringBooksPdf.com

You can modify these values by simply typing in new values, if required.

NZ in the Z- box implies an infinite safe volume below the component so that the probe can never travel below the part.

The ‘Measure’ button will automatically calculate the safe volume from the features that have been measured in the current inspection. In effect this button will perform the same calculation as the calculation performed when prompted to automatically calculate it when running a program for the first time.

www.EngineeringBooksPdf.com

28.0 SOFTWARE SET UP

Many of the parameters used by the software may be set by the user to suit their particular requirements. In order to change any of these parameters click on the Set Up button:

Fig. 28.0-1 The Software Set up button

This will bring up the Set Up window, as follows:

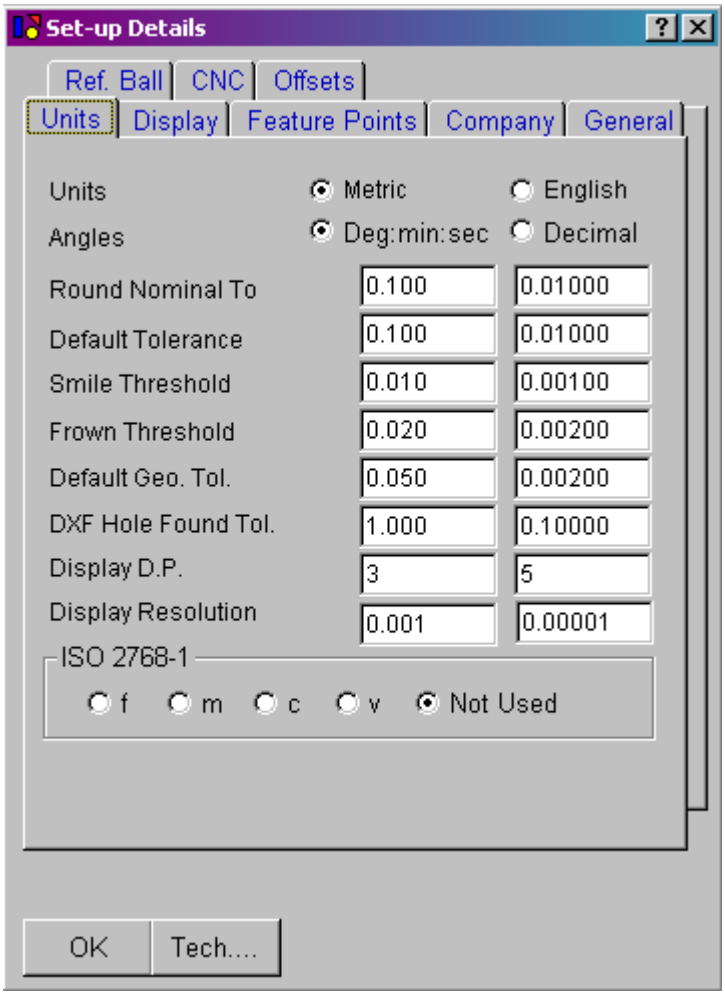


Fig. 28.0-2 User Definable Parameters - Units

Clicking on the heading tabs will bring up each of the lists of user definable parameters. To change any of the parameters simply type in the new value over the existing. If you wish to use the new value simply for the current inspection, and revert to the saved value next time the software is used, then click only on ‘OK’. If however you wish to save the new value, and use it until such times as it is modified again, then click on the ‘Save’ button followed by ‘OK’

www.EngineeringBooksPdf.com

The ‘Tech….’ Button will allow access to more technical information such as the machine’s error map. To bring up this information a code must be typed in. This code is available from Aberlink for servicing and problem solving only.

28.1 Units Tab

This is the information shown in the figure above (Fig 28.0-2) The 2 columns of boxes allow different values to be entered when working in metric values compared to when working in English values:

28.1.1 Metric or English

Use the option buttons to select whether the results are given in millimetres or inches.

28.1.2 Display Angles as Deg, Min, Sec or as a decimal

Use the option buttons to select whether angles are reported as degrees, minutes and seconds or decimal angles. When changing between the two options the screen will need to be refreshed before showing the new angle information. The Redraw function will do this

28.1.3 Round Nominals To

When printing out inspection reports of Tabulated Dimensions, it is possible to print out a list of nominal dimensions alongside the measured values, and hence also produce values for the measured errors.

The nominal values may be entered by right mouse clicking on the dimension, to bring up the dimension detail box – see chapter 27, Printing Results.

If however, no nominal dimension has been entered for a given dimension, then the default nominal will be the printed as the measured value rounded to the precision set in this box, and the tolerance on the dimension will be set to plus or minus half this rounding value.

28.1.4 Default Tolerance

Unless otherwise specified the software will apply +/-the value given as the tolerance for each dimension. To modify the tolerance for a specific dimension right mouse click on the dimension line, to bring up the dimension detail box

www.EngineeringBooksPdf.com

28.1.5 Smile / Frown Threshold

When measuring any type of feature there are 2 levels of warning that are given to let the operator know if a particular measurement point does not fit the form of the feature very well. The thresholds at which these warnings are given can be set in these boxes The Smiley Face feature will remain smiling providing that the error of the last measurement point does not exceed the threshold value set in the Smile Threshold box.
If the error of the last measurement point falls between the smile threshold and the frown threshold the smiley face feature will have an open mouth (frown). If the last measurement point error exceeds the frown threshold then the Smiley Face feature will become unhappy and an audible warning will be given.

28.1.6 DXF Hole Found Tolerance

It is possible to import hole positions for a component in a DXF file format. When doing this the software will intelligently pair the measured results to the positions defined in the DXF file. In this case the software will use the tolerance set in this box to determine whether a measured hole is paired with inputted values.

28.1.7 Default True Pos. Tolerance

When printing point position reports (see chapter 27 – Printing Results) the true position of points and circles can be printed. This box can be used to set a default tolerance for true position values, which can then be used to produce a Pass/Fail statement.

28.1.8 Display D.P.

Use this box to modify the number of decimal places shown on the readouts, measured dimensions and in all inspection reports.

28.1.9 Display Resolution

This box is used to modify the display resolution if you enter 0.001 the display will count 1 micron steps, if you enter 0.005 the display will count in 5 micron steps.

28.1.10 The ISO 2768 Tolerance standard

You can use the ISO 2768 tolerance standard to tolerance your parts by clicking on the buttons marked f,m,c & v the software will than apply the correct tolerances take from the tables below to the parts you are measuring.

www.EngineeringBooksPdf.com

Table 1 - Permissible deviations for linear dimensions except for broken edges (external radii and chamfer heights)

Tolerance class		Permissible deviations for basic size range							
Designation	Description	from 0,5* up to 3	over 3 up to 6	over 6 up to 30	over 30 up to 120	over 120 up to 400	over 400 up to 1000	over 1000 up to 2000	over 2000 up to 4000
f	fine	± 0,05	± 0,05	± 0,1	± 0,15	± 0,2	± 0,3	± 0,5	--
m	medium	± 0,1	± 0,1	± 0,2	± 0,3	± 0,5	± 0,8	± 1,2	± 2
c	coarse	± 0,2	± 0,3	± 0,5	± 0,8	± 1,2	± 2	± 3	± 4
v	very coarse	--	± 0,5	± 1	± 1,5	± 2,5	± 4	± 6	± 8
* For nominal size below 0,5 mm, the deviation shall be indicated adjacent to the relevant nominal size(s).									

Table 3 – Permissible deviations for angular dimensions

Tolerance class		Permissible deviations for ranges of lengths in millimetres of shorter side of the angle concerned				
Designation	Description	up to 10	over 10 up to 50	over 50 up to 120	over 120 up to 400	over 400
f	fine	± 1°	± 0,5°	± 0,333°	± 0,166°	± 0,083°
m	medium					
c	coarse	± 1,5°	± 1°	± 0,5°	± 0,25°	± 0,166°
v	very coarse	± 3°	± 2°	± 1°	± 0,5°	± 0,333°

Indication on drawing – example (medium): ISO 2768-m alternatively SS-ISO 2768-1 m

www.EngineeringBooksPdf.com

28.2 Display Tab

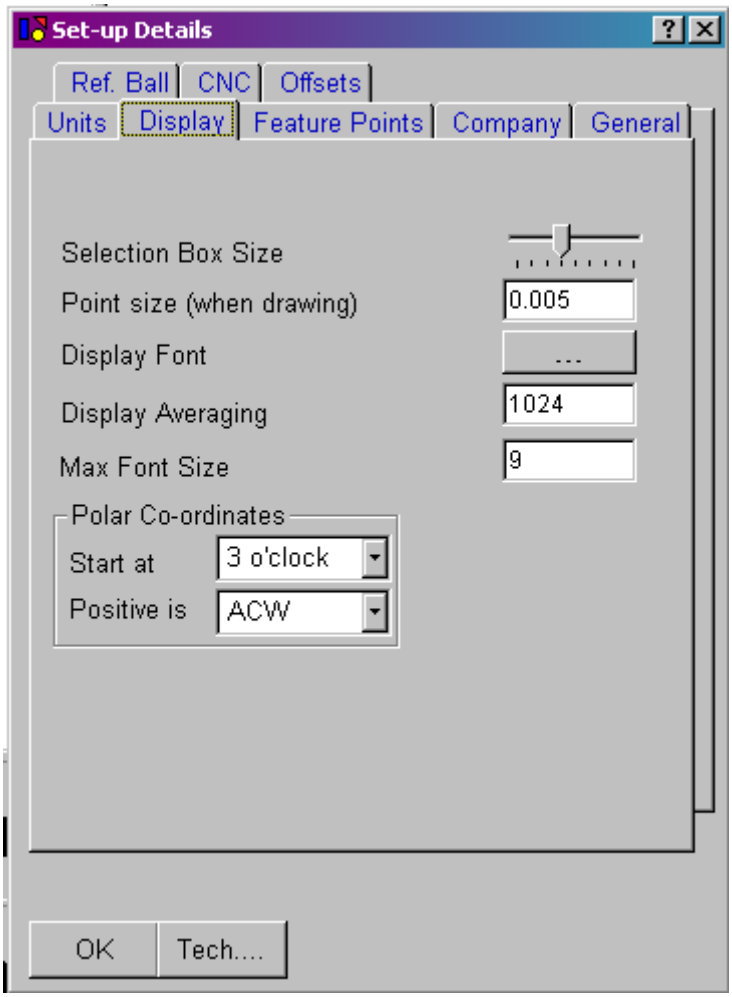


Fig. 28.2-1 User Definable Parameters – Display

28.2.1 Selection Box Size

The slider bar can be used to adjust the size of the selection box, that appears in the graphical areas of the Main Screen.

28.2.2 Point Size (when drawing)

When measuring or constructing points, they appear in the Main Screen as blue filled circles. Modifying the value in this box will alter the size of these points.
Bigger points are easier to select small look better in the reports.

28.2.3 Display Font

www.EngineeringBooksPdf.com

The font, style and size of the results can be selected by clicking on the grey box. This will bring up the following window:

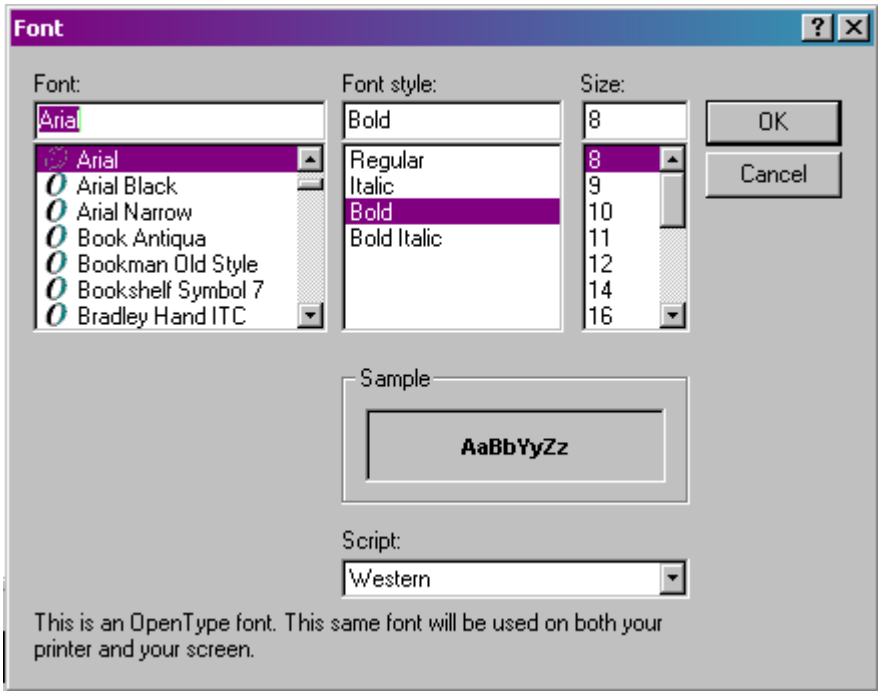


Fig. 28.2-2 The Font window

The font, style and size of the printed results may now be selected from the lists available within Windows XP.

28.2.4 Max Font Size

If the font size is bigger than 9 the letters will be to big to fit the buttons or drop down menus

28.2.5 Display Averaging

This box affects the stability of the readouts. The software will take this number of position readings from the readheads on the machine (max 1024) before displaying the average. The higher this number the more stable the readings will appear, hence this facility will control the flicker of the readouts.

28.2.6 Polar Co-ordinates

BS 308 defines the angle of a polar co-ordinate as 3 o'clock on a clock face equating to zero degrees, and anti-clockwise defining a positive angle (hence 12 o'clock = 90 degrees and 6 o'clock = 270 degrees etc.)
If you wish to work to an alternative standard you can use these boxes to define zero degrees and which direction defines a positive angle.

www.EngineeringBooksPdf.com

28.3 Feature Points

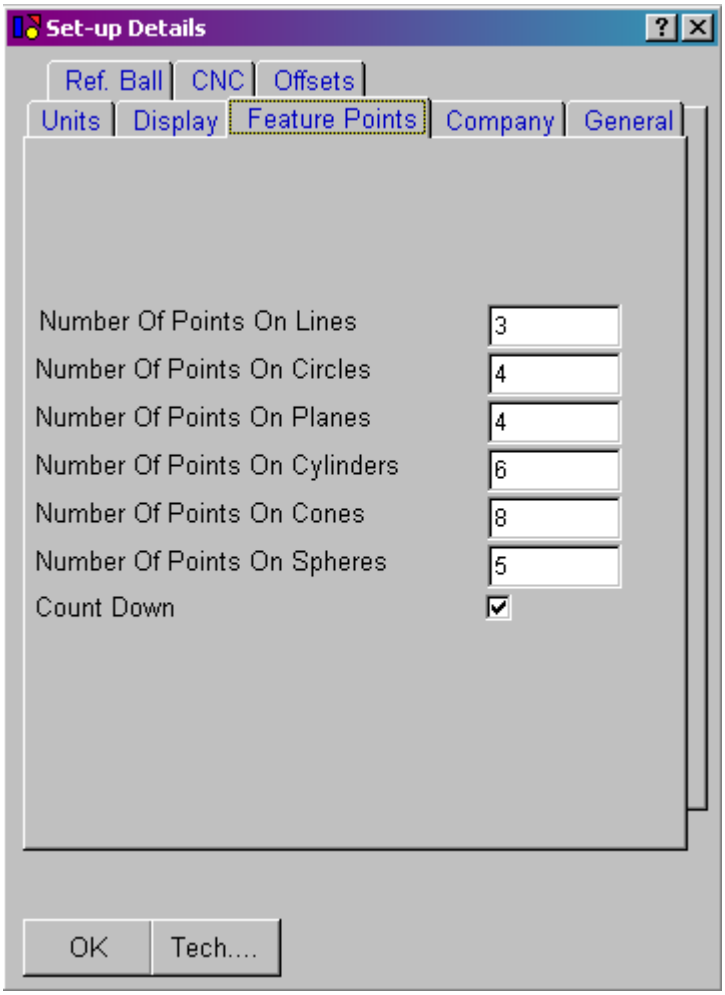


Fig. 28.3-1 User Definable Parameters – Feature Points

During normal operation, features are inspected by the operator taking an unspecified amount of measurement points, and the software will count these points upwards from zero. It is up to the operator to decide when they have taken sufficient points.

If however, the Count Down box in this window is ticked, then the measurement windows will start to count down from the specified number of points defined in the boxes above.

www.EngineeringBooksPdf.com

28.4 Company

Fig. 28.4-1 User Definable Parameters – Company

The name and address shown in this window will appear on all printed inspection reports.

www.EngineeringBooksPdf.com

28.5 General

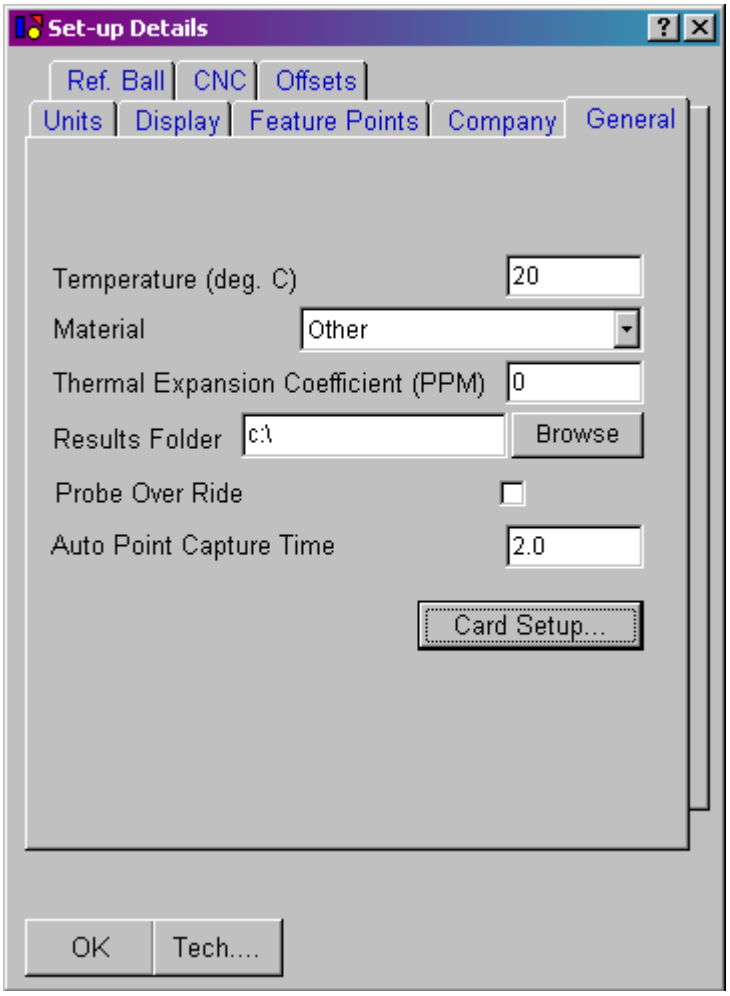


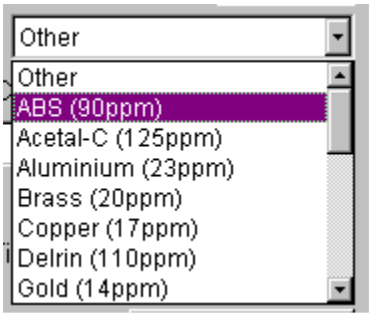
Fig. 24.5-1 User Definable Parameters – General

28.5.1 Temperature (deg C)

Most CMM structure are made from a single material like Aluminium, it will expand and contract uniformly with temperature. It is therefore possible to compensate for the ambient temperature of the machine in the software. This value should therefore represent the mean ambient working temperature, and may for instance be changed between the winter and the summer.

www.EngineeringBooksPdf.com

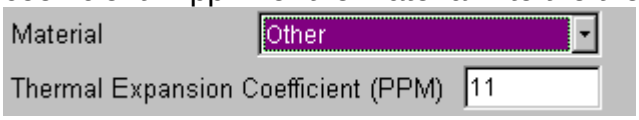
28.5.2 Material



Different materials expand and contract at various rates as the temperature changes, by selecting from the list of materials shown in the ‘Matrial drop down menu’ the material your measured part is made from, you can compensate for that part expand and contract as the ambient working temperature changes over time.

28.5.3 Thermal Expansion coefficient (PPM)

Most of the materials you are ever going need are listed in the ‘Materials Drop down menu’ if they are not listed you can select ‘other’ from drop down menu After you have selected ‘other’ you must enter the thermal expansion coefficient in ppm for the material into the thermal expansion coefficient box.



If you have selected a specific material from the ‘Materials Drop down menu’ then its expansion coefficient is stored in the software and will have automatically been entered.

28.5.4 Results Folder

The path shown in this box will be the default path called up by the software when saving results using the Save File and Save As buttons.

This path can of course be changed each time an inspection is saved, but if inspection results are generally saved to a specific results folder, then this results folder path should be entered here.

In the example above, the Save File and Save As buttons would by default bring up the root directory C:\

28.5.5 Probe Override

Ticking this box will have the effect of ignoring the signal from the probe. This may be useful during faultfinding etc. when the probe may need to be bypassed.

This box would also need to be ticked if a solid probe were ever used in the

www.EngineeringBooksPdf.com

place of the touch trigger probe.

28.5.6 Auto Point Capturetime

If you are using a solid probe on a manual machine to measure profiles by dragging the probe across the surface it is possible to capture points at set intervals over time.
You enter the time between points here.

28.6 Ref Ball Tab

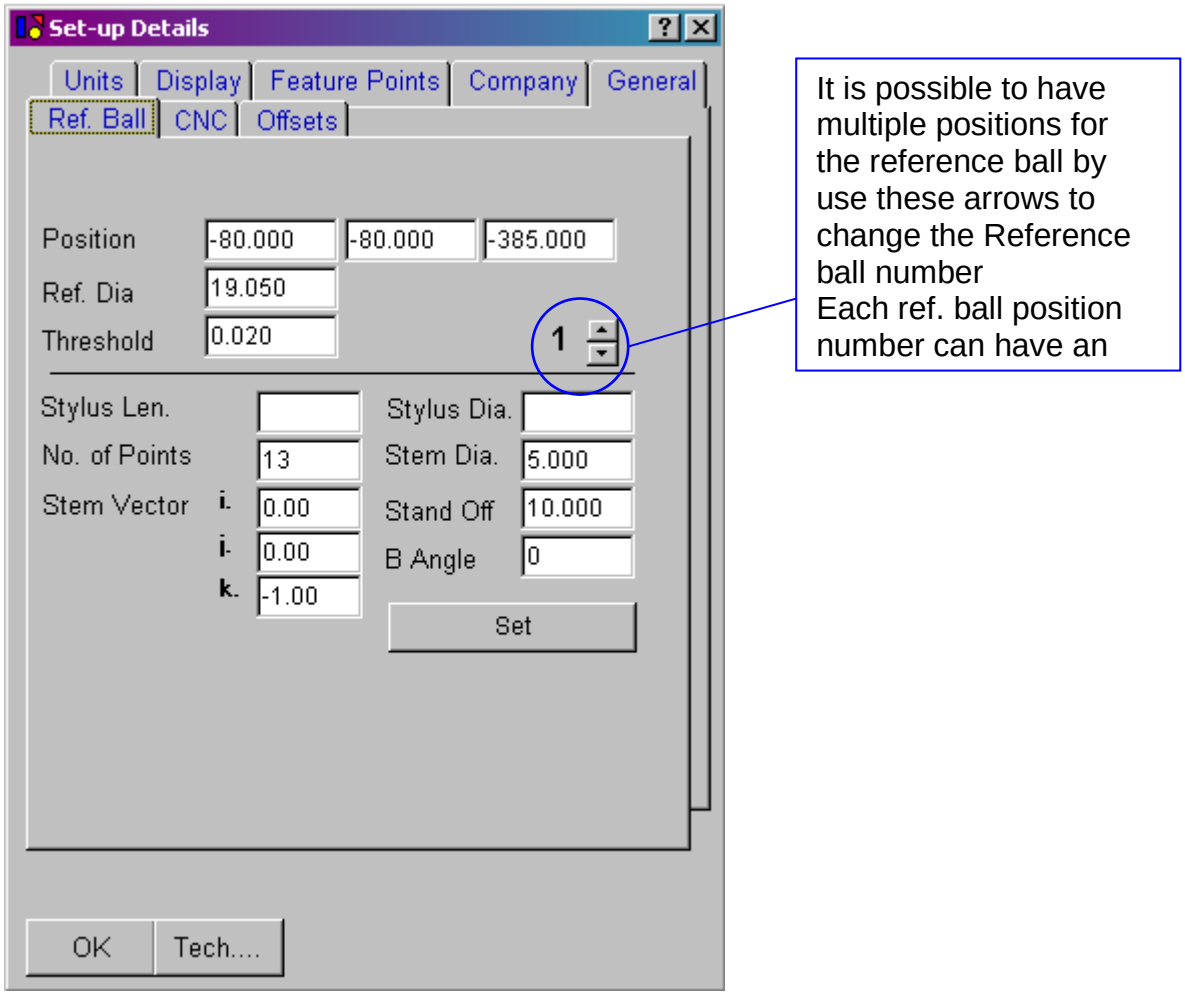


Fig. 28.6-1 User Definable Parameters- Ref. Ball

28.6.1 Reference Ball

The 3 boxes denote the position of the reference ball in the measurement volume of the machine relative to the machine's datum position (top, back left corner). If the reference ball is moved to another position within the measurement volume its position must be entered here. The procedure for measuring the reference ball position is detailed in chapter….
Note:- If you move the reference ball to different place on the machine table it is import to re-enter the new position in to the X,Y,Z boxes relative to the

www.EngineeringBooksPdf.com

machine's datum position If you are running an error map on your machine and you fail to do this you will get measurement errors.

28.6.2 Ref. Dia
The diameter of the reference ball must be entered here.

28.6.3 Threshold
This box allows the user to set a threshold for the 3 sigma value of the points fit when Datuming a stylus. If the 3 sigma value should exceed this threshold value then the appropriate box on the stylus Datuming window will turn yellow, rather than green or red.

28.6.4 No. of points
In this box you can enter the number of point you need to take when calibrating your probes

28.6.5 Stylus Len. and Stylus Dia.
The approximate length and diameter of the stylus used when measuring the reference ball position should be entered into these boxes. The procedure for measuring the reference ball position is detailed in chapter….

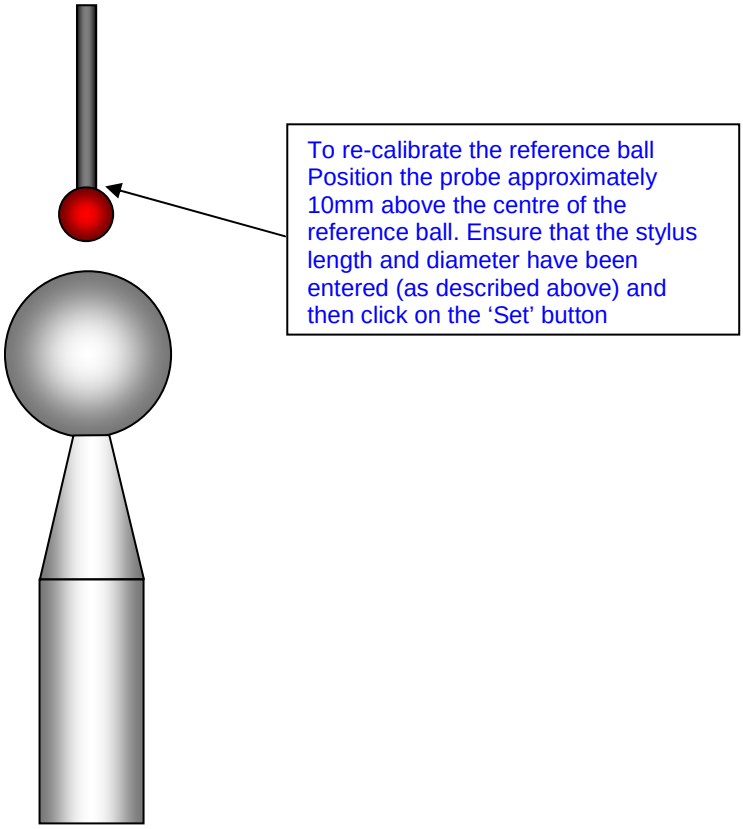
28.6.6 Stem Vector and Stem Dia.
The stem vector tells the software which from which direction the reference ball is attached to its pillar (0,0,-1 is straight down). The stem diameter is the diameter of the pillar that the probe must avoid when Datuming a stylus.

28.6.7 Stand Off
Defines the pre-travel for each measurement point when Datuming a stylus.

28.6.8 B Angle Set
This button is used for automatically measuring the reference ball position. This should only be done if the reference ball position has moved within the measurement volume of the machine, for instance been screwed into a different insert or if the granite table has been moved.

www.EngineeringBooksPdf.com

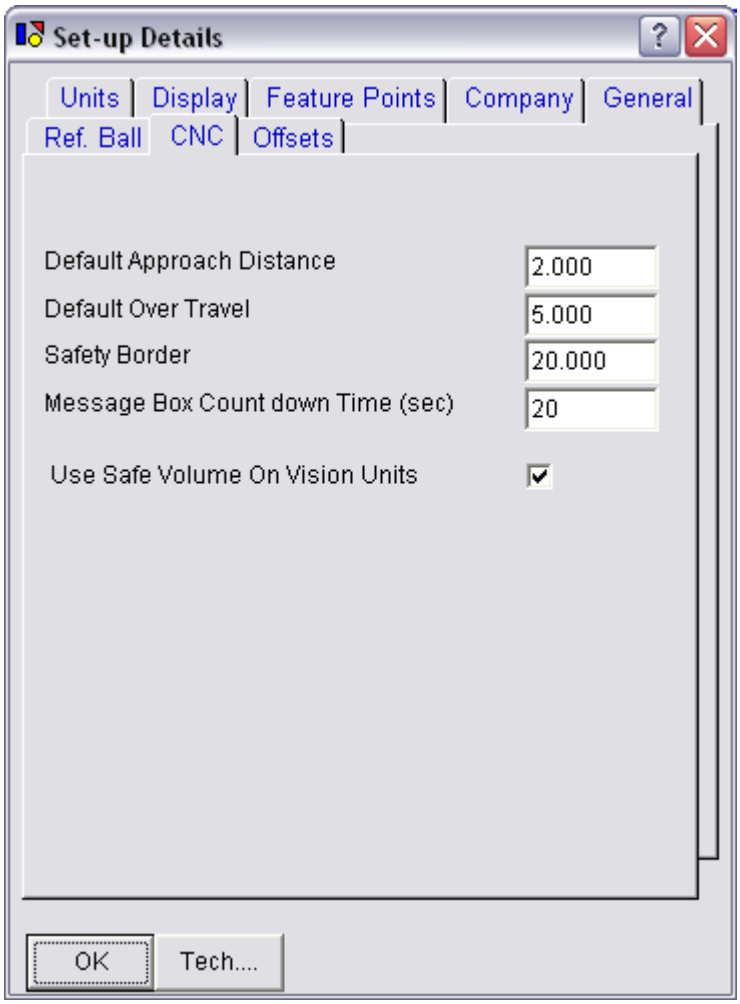
28.6.9 How to re-calibrate the reference ball position.



To do this position the probe approximately 10mm above the centre of the reference ball. Ensure that the stylus length and diameter have been entered (as described above) and then click on the 'Set' button. The machine will now automatically measure the position of the reference ball. If an indexible probe is fitted the probe will also index through 90 degrees and measure the reference ball a second time. The X,Y,Z position of the ball will automatically be entered at the top of the screen. Be sure to click 'save' to permanently save these values before clicking OK.

www.EngineeringBooksPdf.com

28.7 CNC Tab



28.7.1 Default Approach Distance

Defines the default value for pre-travel used when creating a program. Pre-travel is the distance from a theoretical measurement point to which the probe will travel at rapid speed. A small pre-travel will increase the speed of measurement but a large pre-travel will allow more uncertainty of positioning of a component on the table.

28.7.2 Message Box Count down Time

This is the time a message box is displayed in seconds

28.7.3 Default Over Travel

Defines the default value for over travel used when creating a program. Over travel is the distance past a theoretical measurement point to which the probe will continue travelling at probing speed until the program will give up trying to measure a point and register an error.

Safety Border

If you ask the software to automatically calculate the Safe Volume for a program it will create a volume around all of the measured features of this size in X,Y and Z through which the probe must not travel between measuring different features. The safe volume is always considered to be infinite in a –Z

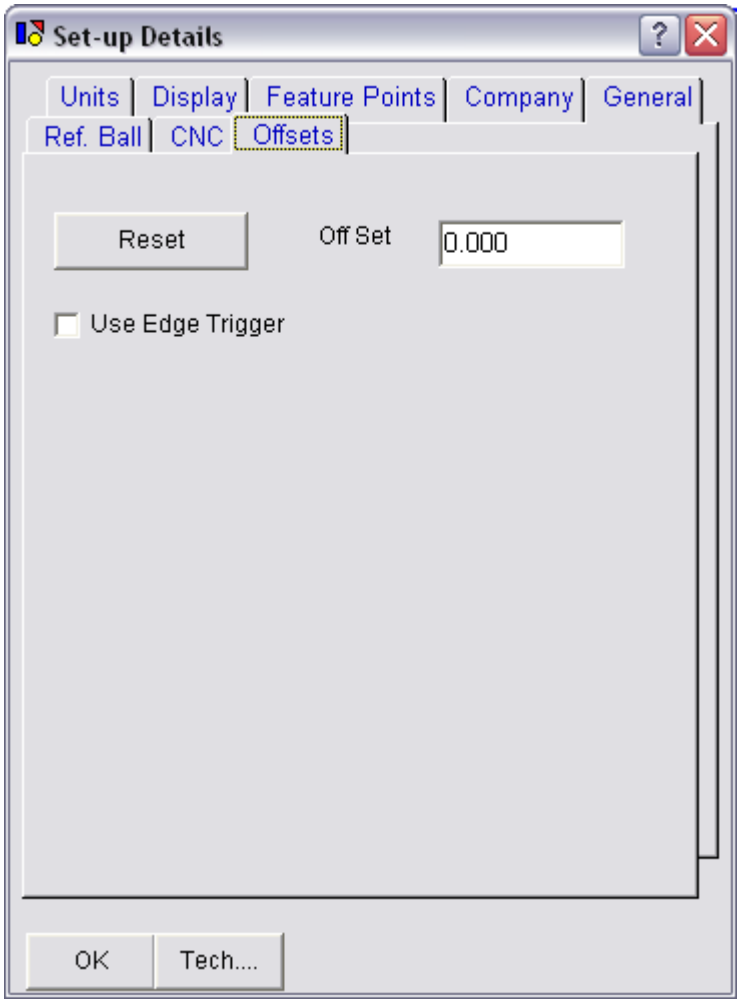
www.EngineeringBooksPdf.com

direction, ie. the probe can never travel underneath the component being measured.

27.7.4Message Box Count down Time

This is the time a message box is displayed in seconds

28.8 Offsets Tab



www.EngineeringBooksPdf.com

Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

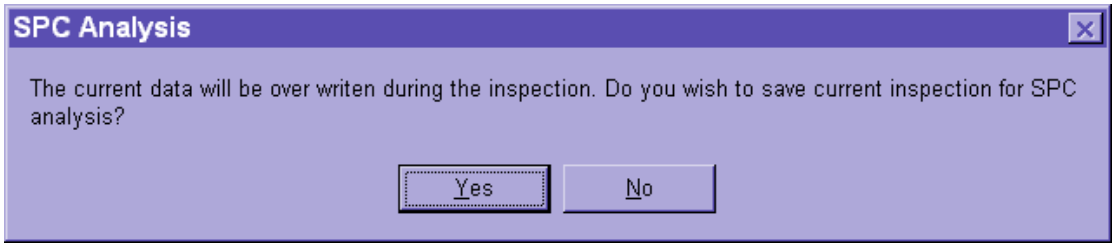
www.EngineeringBooksPdf.com

29.0 SPC Batch information

The Aberlink 3D software will collect batch information as well as calculate some simple SPC information and plot both a histogram and a scatter graph for any given dimension.

In order to collect this information when running a batch of components you need to tick the 'SPC All Components' box on the CNC Program Window (see Chapter – The Play Function).

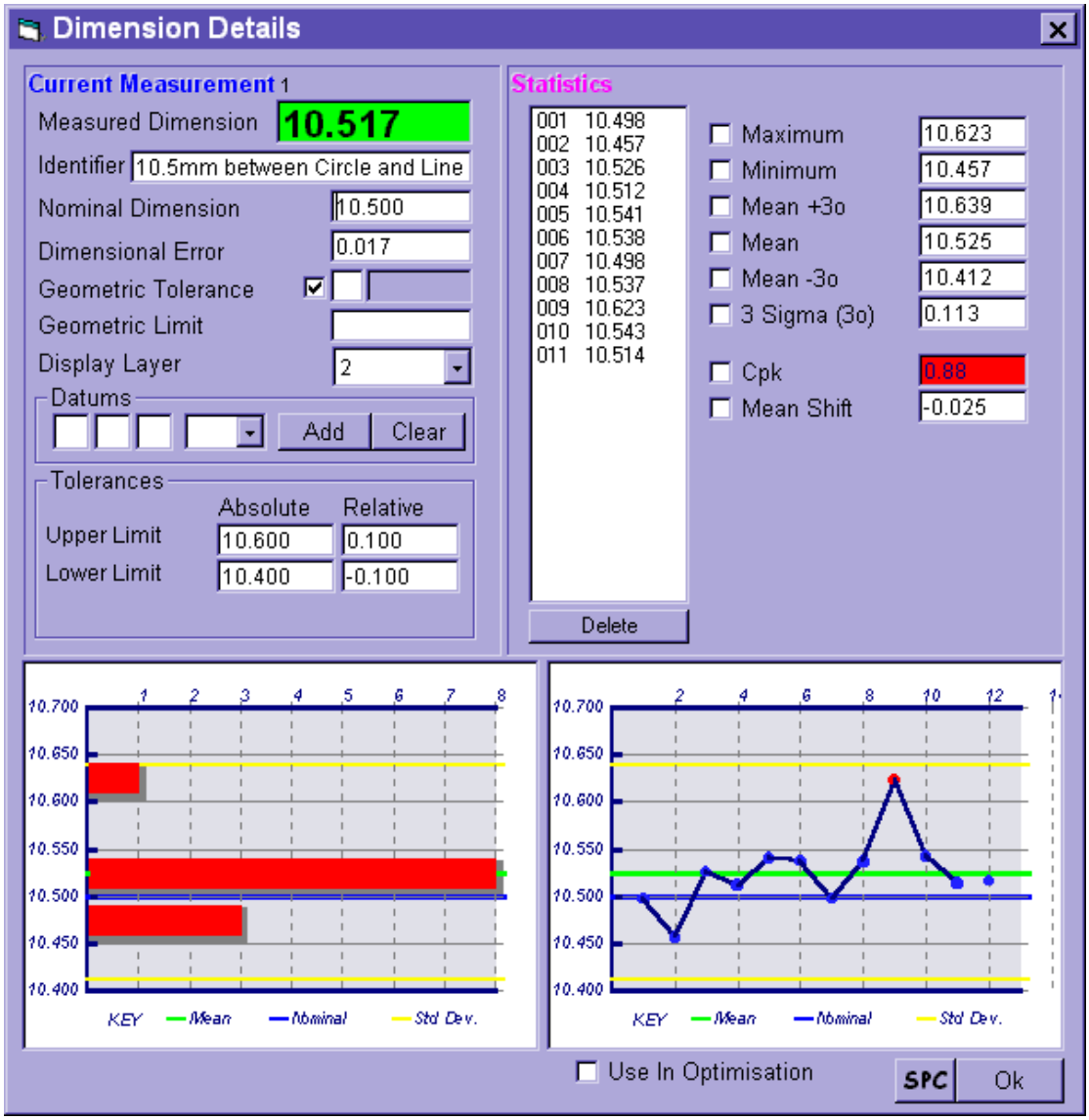
Now when you click on 'Run Program' you will get the following message box:



If you wish to add the previous inspection to the batch information click on 'Yes', otherwise click 'No'.

The batch information is contained within the Dimension Details box. To look at it right click on a particular dimension line. For instance if we select a dimension between a hole and the edge of a workpiece that is nominally 10.5 mm:

www.EngineeringBooksPdf.com



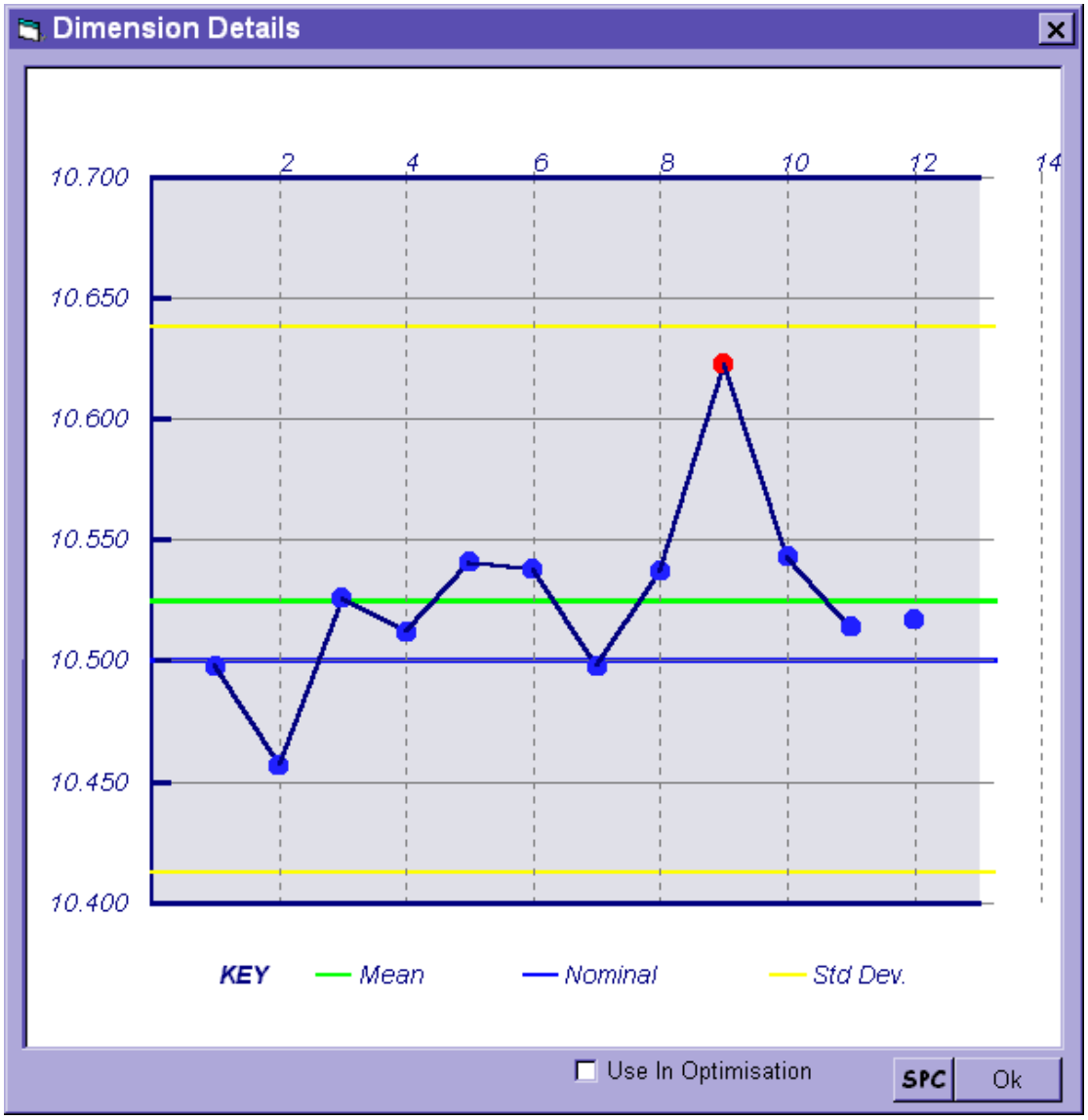
The last measured result is shown large numbers under ‘Current Measurement’ and the dimension for previously measured components are listed under ‘Statistics’.

For the batch the software calculates the following:

1. The Maximum value
2. The Minimum value
3. The Mean +3 sigma value
4. The Mean value
5. The Mean –3 sigma value
6. The 3 sigma value for the batch
7. The CpK value for the batch
8. The Mean shift for the batch (mean – nominal)

In addition the software plots a scatter graph and a histogram for the batch results. To increase the size of these graphs simply click on them.

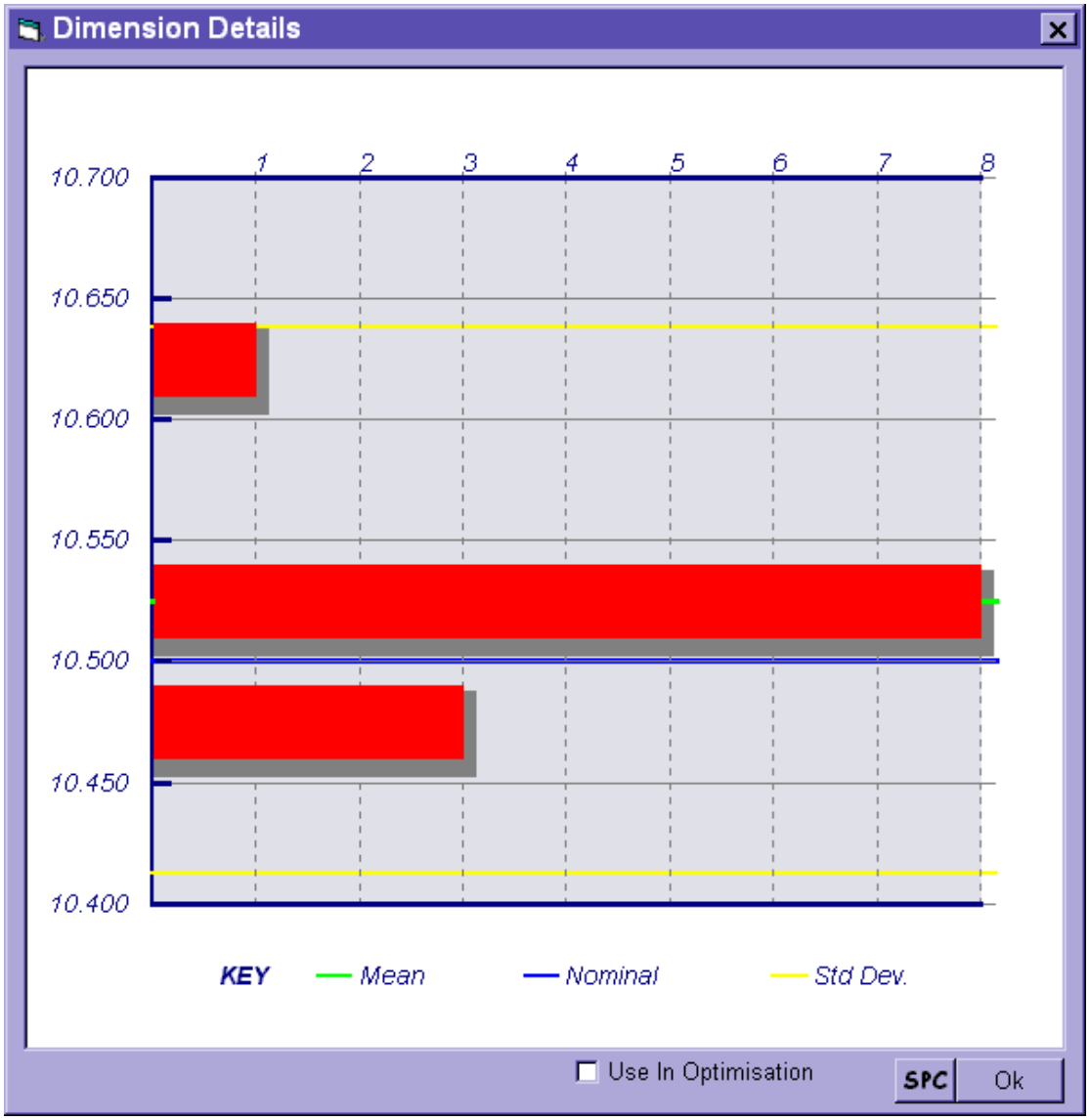
www.EngineeringBooksPdf.com



The scatter graph will show the mean, the nominal and the plus and minus sigma values. Measurements within limits are shown as blue dots, and out of limits are shown as red dots.

To return to the original view simply click on the graph again. To expand the histogram just click on it.

www.EngineeringBooksPdf.com



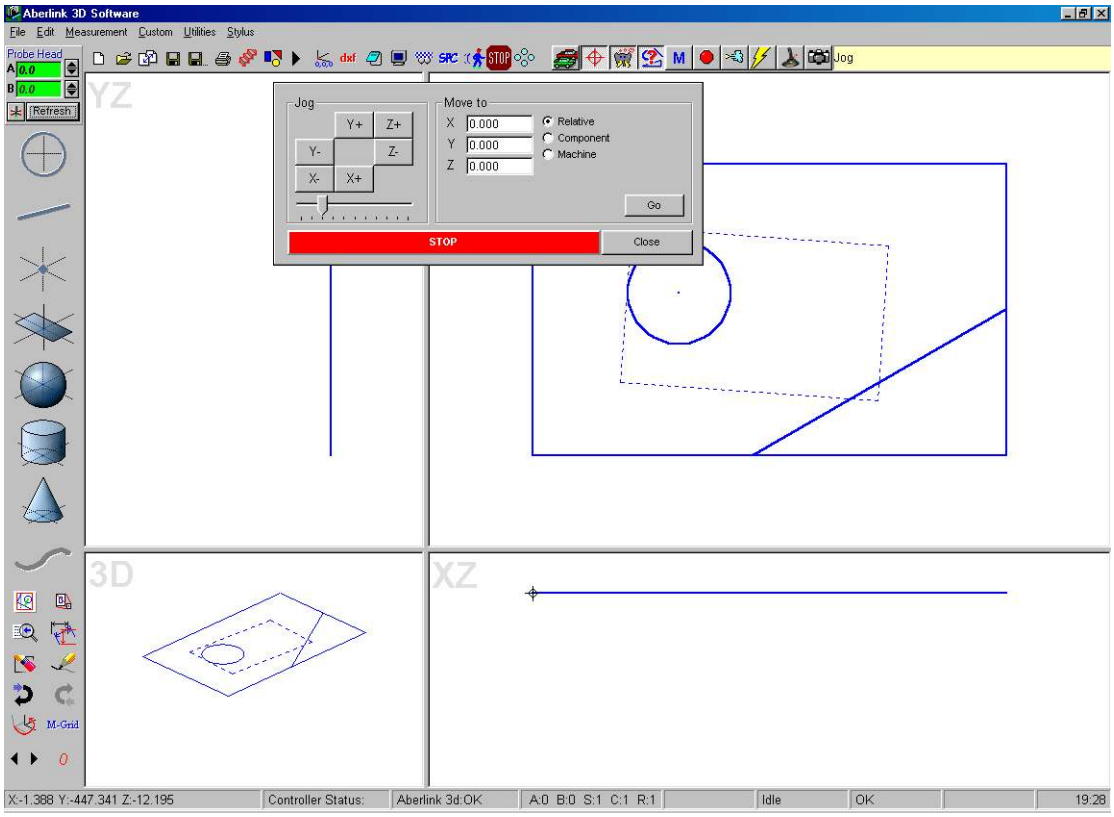
Again the mean, the nominal and the plus and minus sigma values are shown. To return to the original view click on the histogram again. To return to the main screen click ‘OK’.

www.EngineeringBooksPdf.com

30.0 THE JOG MACHINE BUTTON

Should the joystick become broken for any reason the machine can still be driven around using the 'Jog' function.

This will bring up the following window:



The machine can now be moved around by left clicking on the appropriate direction box. Note that speed can also be controlled by using the slider bar at the bottom of the window.
Right clicking on a direction button will produce a probing move equivalent to using buttons 1,3 and 4 on top of the joystick.

The Jog function will also allow the user to move to a defined position using the 'Go To' boxes. Type in the desired X,Y,Z position and also select whether this is:

1. Relative - Relative to the current position of the probe

2. Component - Relative to the datum on the component being measured

3. Machine - Relative to the machine's datum position

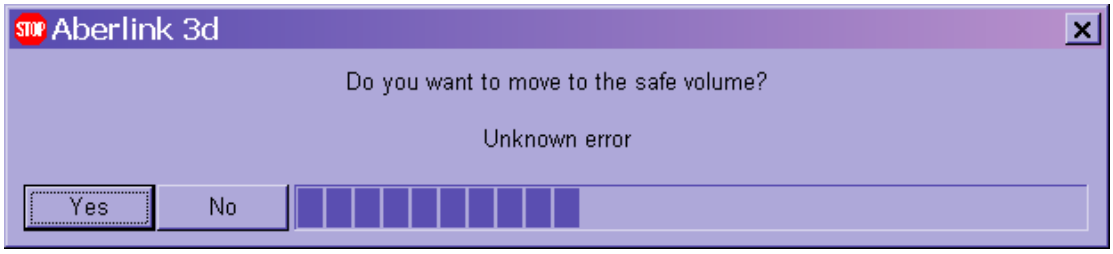
Then click on 'Go To'.
Finally, there is a 'Stop' button to halt the travel at any point if required.

www.EngineeringBooksPdf.com

31.0 THE STOP BUTTON

The ‘Stop’ button can be used to halt a program at the end of its current move:

This will bring up the following window:



Then follow the prompts to exit the program.

Note that this button does not perform quite the same function as an emergency stop button on the machine, which will halt all movement immediately and cut the air supply to the machine and disengage the motor drives.

www.EngineeringBooksPdf.com

32.0 THE PARK FUNCTION

The ‘Park’ function will drive the machine to a safe position in the machine volume (usually towards the top, back, right-hand side of the machine) and if an I/O card is fitted to the machine will automatically switch off the air and drives in readiness for the controller to be switched off after use.

When you click on the ‘Park’ button the machine will move upwards in Z first before moving in X and Y, so make sure that the path is clear prior to using this function.

When the machine reaches the safe position the air and drives will be switched off automatically.

This position will be suitable to start from when referencing the machine next time that it is switched on – see chapter 1 – Starting the Machine.

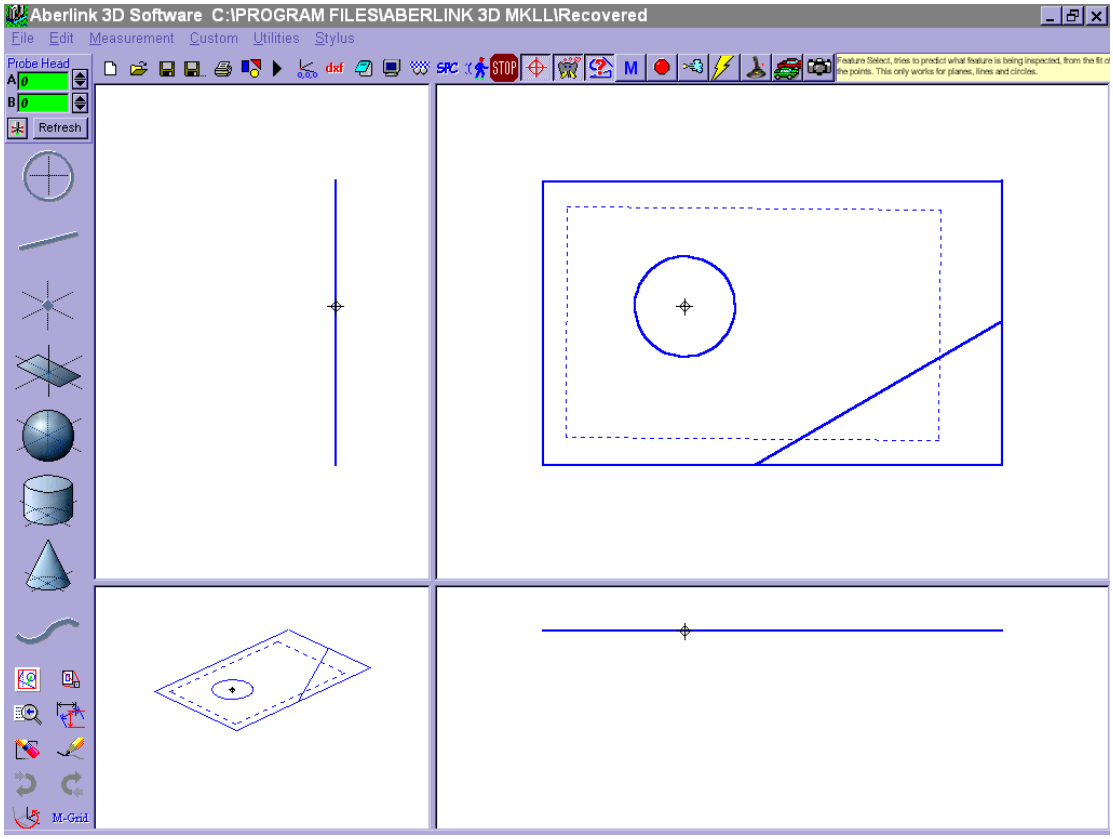
www.EngineeringBooksPdf.com

33.0 DISPLAYING THE PROGRAM TEMPLATE

Each inspection program created in Aberlink 3D can be automatically run. (see Section 27.0 – The Play Function)

In order to create this program the software will use either the features from the very first component measured or alternatively the features downloaded from a dxf file of the component as a template. This template becomes fixed and is used to prompt the measurement of subsequent components.

By default features measured will be shown in blue on the main screen:

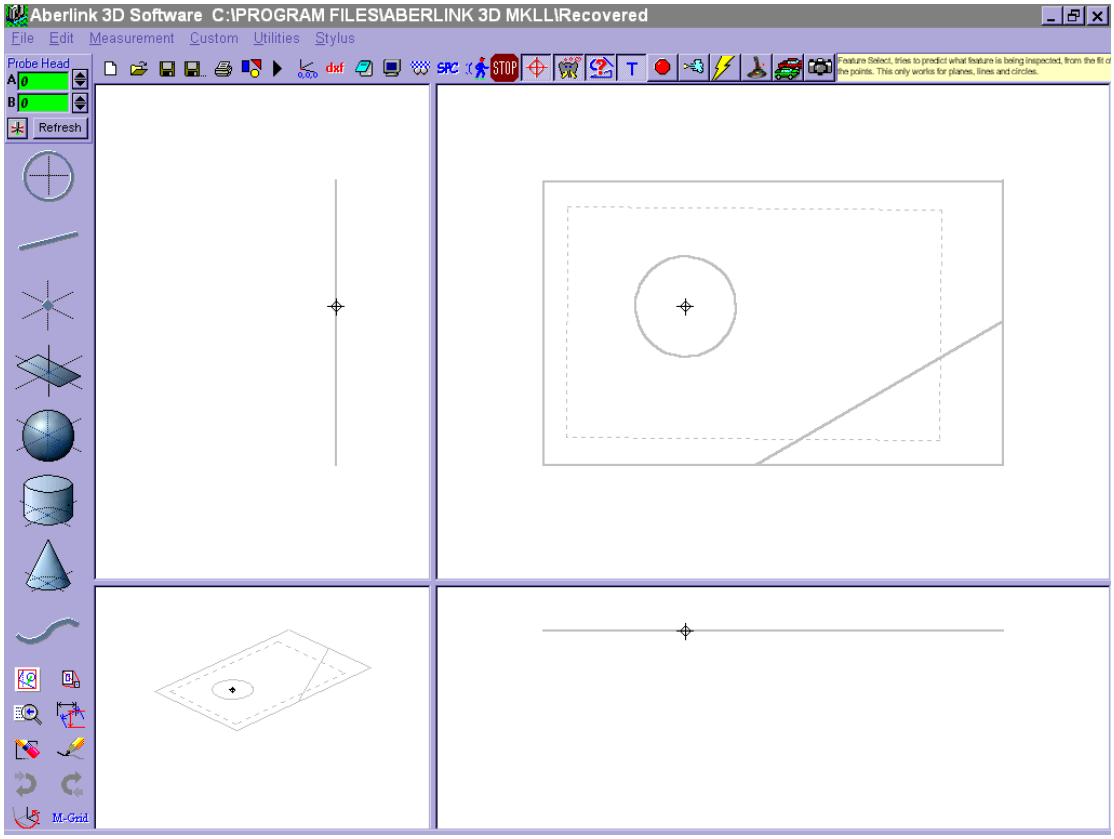


These measured features will be displayed as long as the 'Display Measured Details or Template Details' button is set to 'M', as follows:

If the user wishes to view the template, simply click on this button and it will change to a 'T' for template.

www.EngineeringBooksPdf.com

The main screen will now show the template for the component with the features now depicted in grey:



To return to viewing the measured features simply click on the button again and it will change back to ‘M’.

www.EngineeringBooksPdf.com

34.0 THE AIR SWITCH

The air button is used for switching on and off the air supply as long as an I/O card is fitted to the controller.

Note that the air supply cannot be switched off if the motor drives are engaged. If you try to do this the air supply and motor drives will both be switched off simultaneously.

www.EngineeringBooksPdf.com

35.0 THE MOTOR DRIVES BUTTON

The Motor Drives button is used for engaging and disengaging the motor drives.

Note that the motor drives cannot be engaged if air supply is switched off. If you try to do this, the following window will appear:

*

If you answer ‘Yes’ the air supply and motor drives will both be switched on simultaneously.

If you answer ‘No’ the motor drives will remain off.

www.EngineeringBooksPdf.com

36.0 THE CCD CAMERA SYSTEM BUTTON

The CCD Camera button is situated at the right hand end of the top row of icons on the Main Screen:

If the machine is fitted with the Aberlink Camera system this button will bring up the image from the camera onto the computer monitor and allow non-contact measurement using the Aberlink 3D software.

The use of the camera is detailed in a separate manual.

www.EngineeringBooksPdf.com

37.0 THE RING / GRID ARRAY

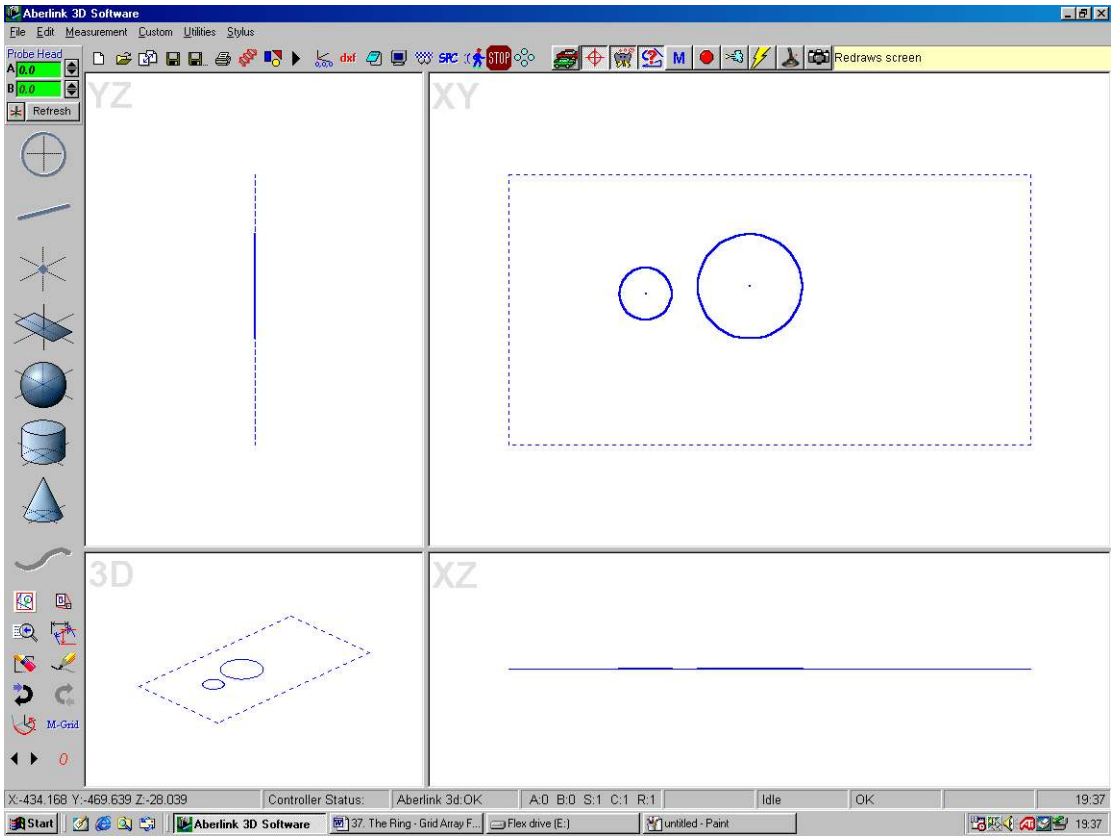
The Ring / Grid array function can be used for the multiple copying of programmed measurement units that form a regular ring or grid pattern

This means for example that if you need to inspect an array of holes using the circle function it is only necessary to create a program for one of them and then use this function to copy the rest of the pattern. Similarly a regular, square or rectangular grid pattern of holes can similarly be measured.

37.1 A Ring Array

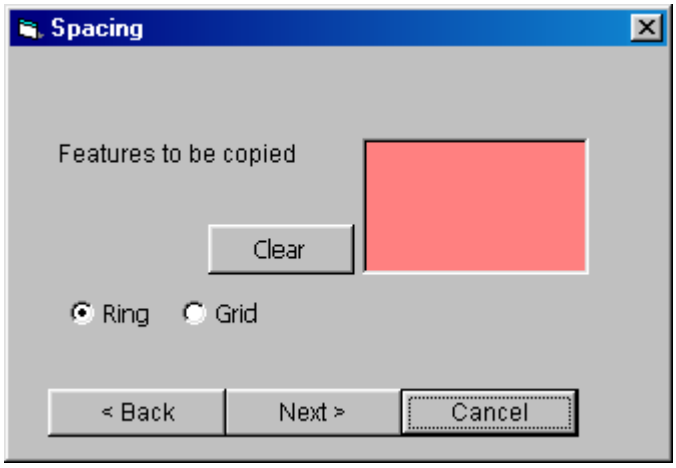
Assume we have a pattern of holes in a surface as follows: 6 holes in a PCD of 24mm, and one hole at the centre of the PCD.

Measure the central hole and also one of the holes in the PCD:

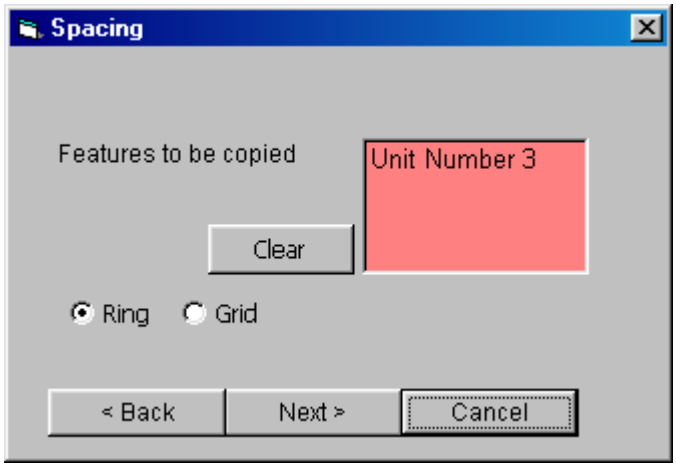


Now click on the Ring / Grid Array function button, which will bring up the following window:

www.EngineeringBooksPdf.com

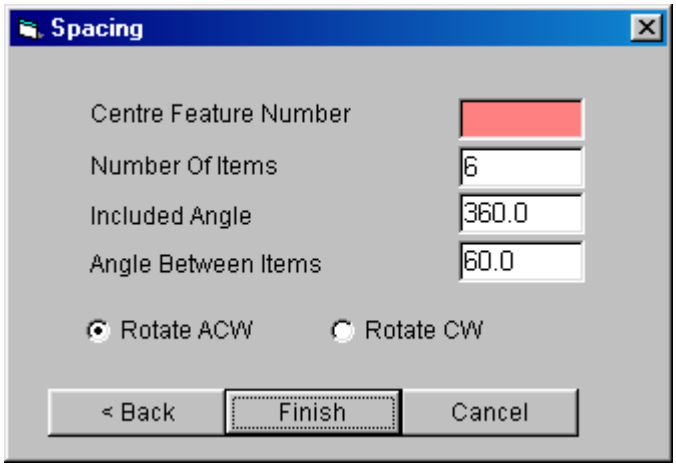


The first box asks for ‘Units to be Copied’. Simply click on the circle that has been measured in the PCD. The unit number of this circle will now be added to the box:



Note that it is possible to select multiple items to be copied, ie. a plane and a circle if measuring holes around the circumference of a cylinder.

Now choose the ‘Ring’ option and click on ‘Next’. The window will now change as follows:



www.EngineeringBooksPdf.com

The first box now asks for the centre of the array. Click on the circle at the centre of the PCD and the unit number of this circle will be added to the box.

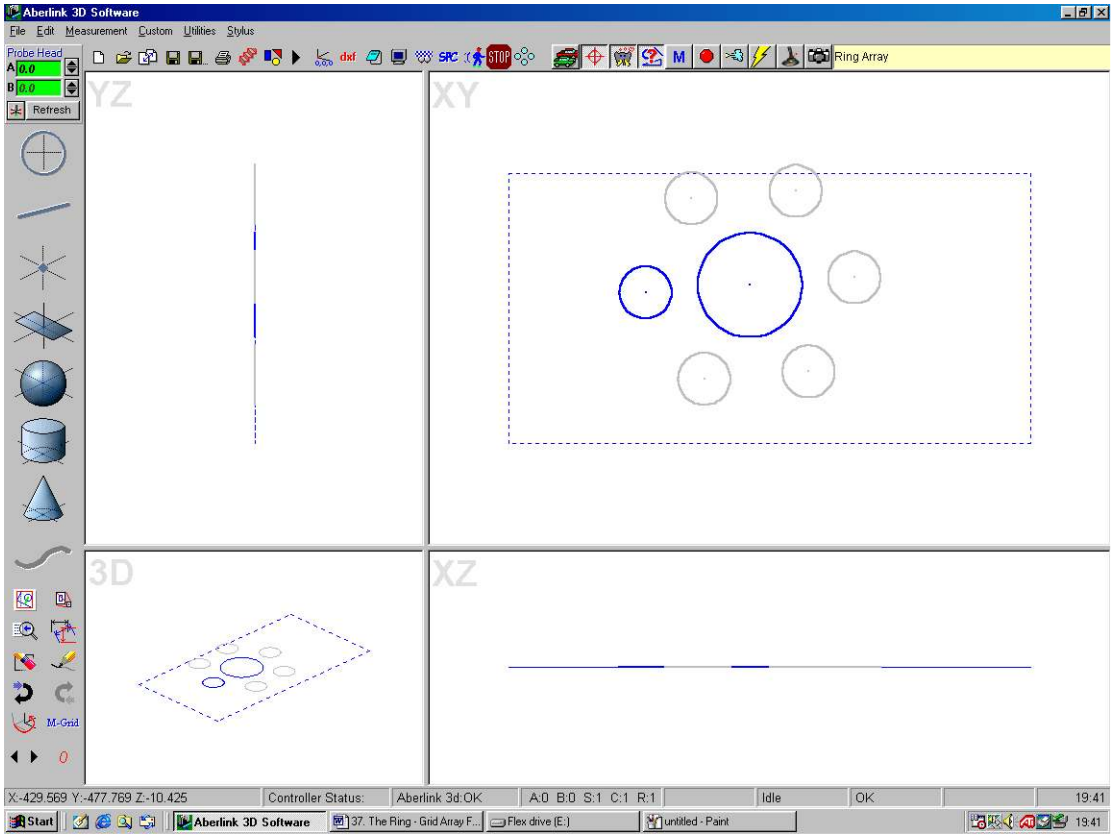
The second box asks for the total number to be copied. In this case enter 6.

The third box asks for the total angle through which we want to copy the circles. As we have a complete PCD we should enter 360.

The last box is the angle between the circles. This will automatically be calculated from the information that we have already given, ie. 6 circles through 360 degrees will be spaced at 60-degree intervals.

Alternatively we could have entered 60 degrees in this box prior to entering the total angle of 360 degrees and the total angle would then have been automatically calculated.

Now click on ‘Finish’ and the remaining 5 circle units will be automatically calculated and displayed as program templates in the main screen:

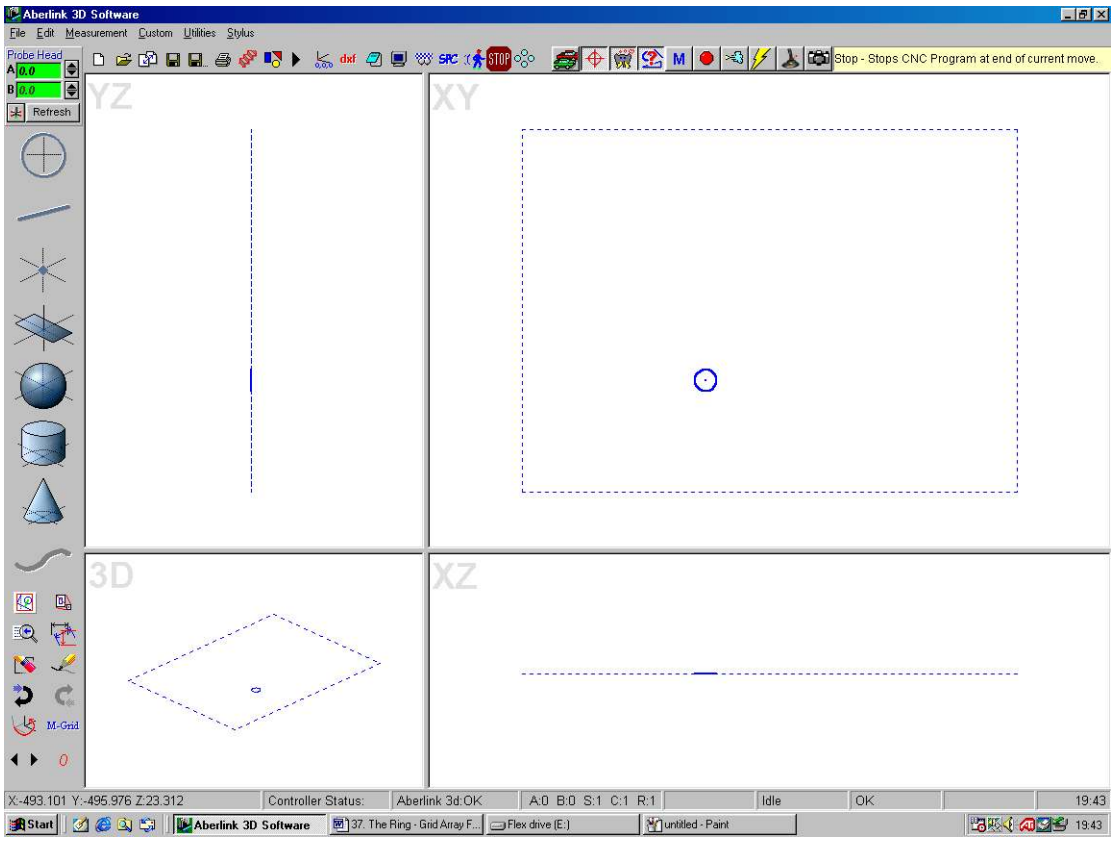


37.2 A Grid Array

It is also possible to copy features in a regular grid pattern (square or rectangular) as follows:

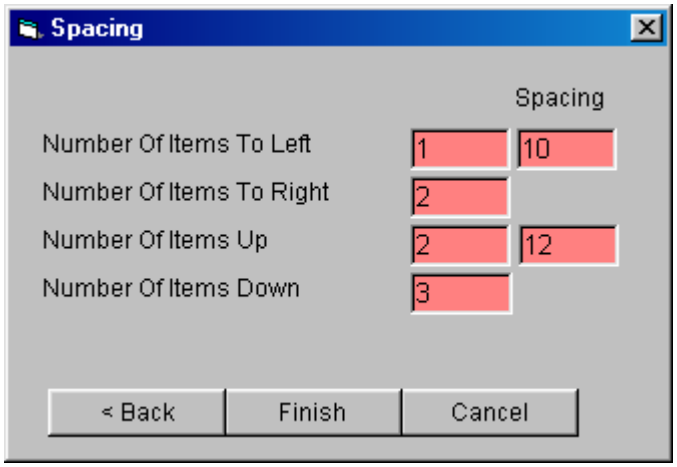
www.EngineeringBooksPdf.com

Measure one of the features in the pattern, for instance we have measured a hole as a circle in the following example:



Now click on the Ring / Grid Array function button and select the circle to be copied, as in the previous Ring Array example.

This time choose the ‘Grid’ option before clicking on ‘Next’. The window will now change as follows:

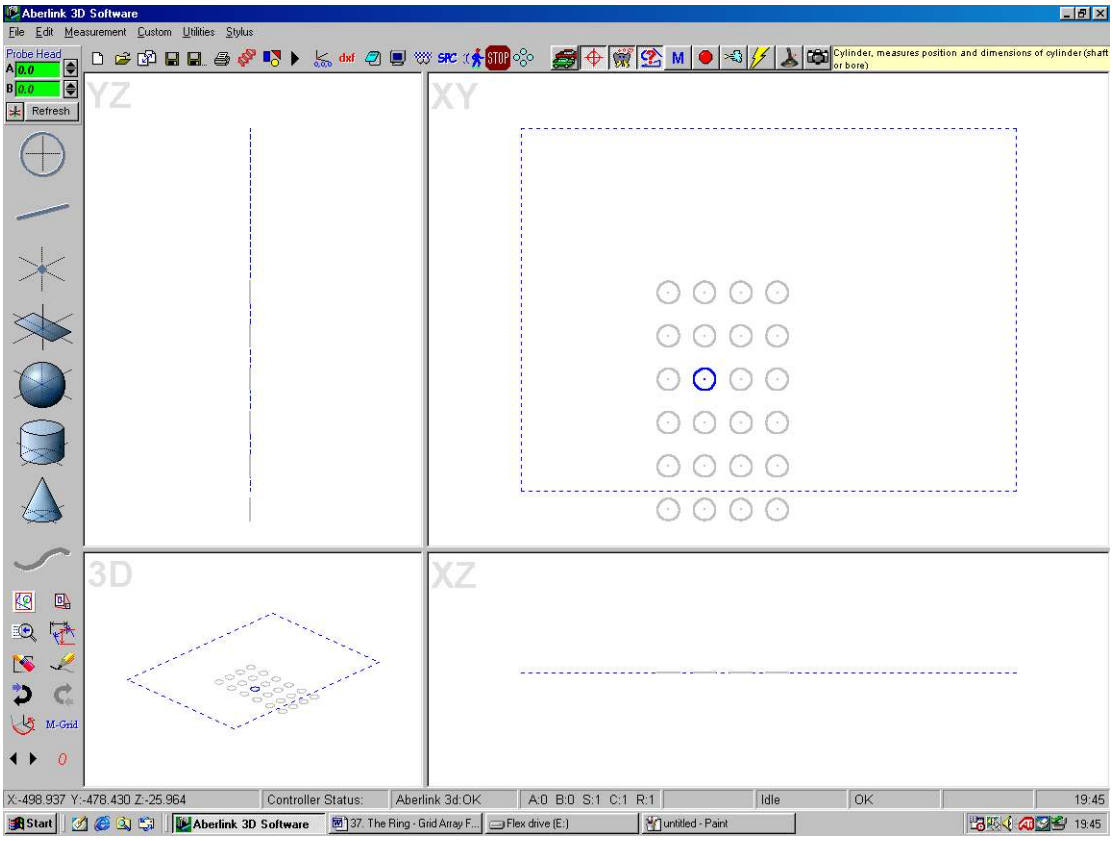


We define the required grid by inputting the number of columns to the left and right of the measured unit together with their spacing, and also the number of rows above and below together with the row spacing.

www.EngineeringBooksPdf.com

In this example enter 1 column to the left and 2 to the right, with a spacing of 10mm, and 2 rows above and 3 below with a12mm spacing. This will create a grid of 4 columns by 6 rows, ie. a total of 24 holes.

Now click on ‘Finish’ and the unmeasured 23 circle units will be automatically calculated and displayed as program templates in the main screen as follows:

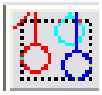


Note that as with the Ring Array function it is also possible to copy multiple units in the same grid pattern.

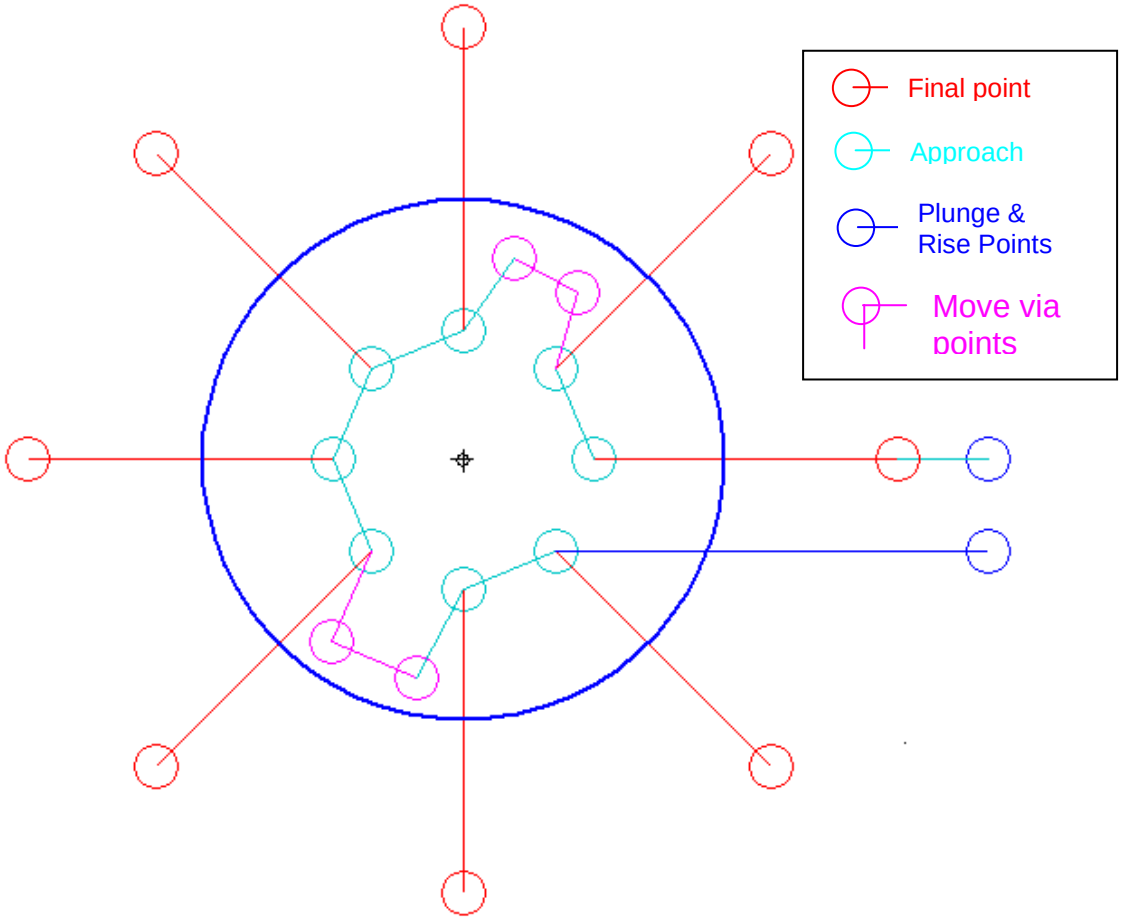
www.EngineeringBooksPdf.com

38.0 Displaying probe contact points

The Aberlink 3D software will allow the user to view on the main screen where measurement points on features have actually been taken. To do this simply click on the 'Display Probe Contact Position' button next left hand click on the feature you wish to see the contact points displayed :



On the main screen the probe contact positions will now be shown as coloured lines & dots:



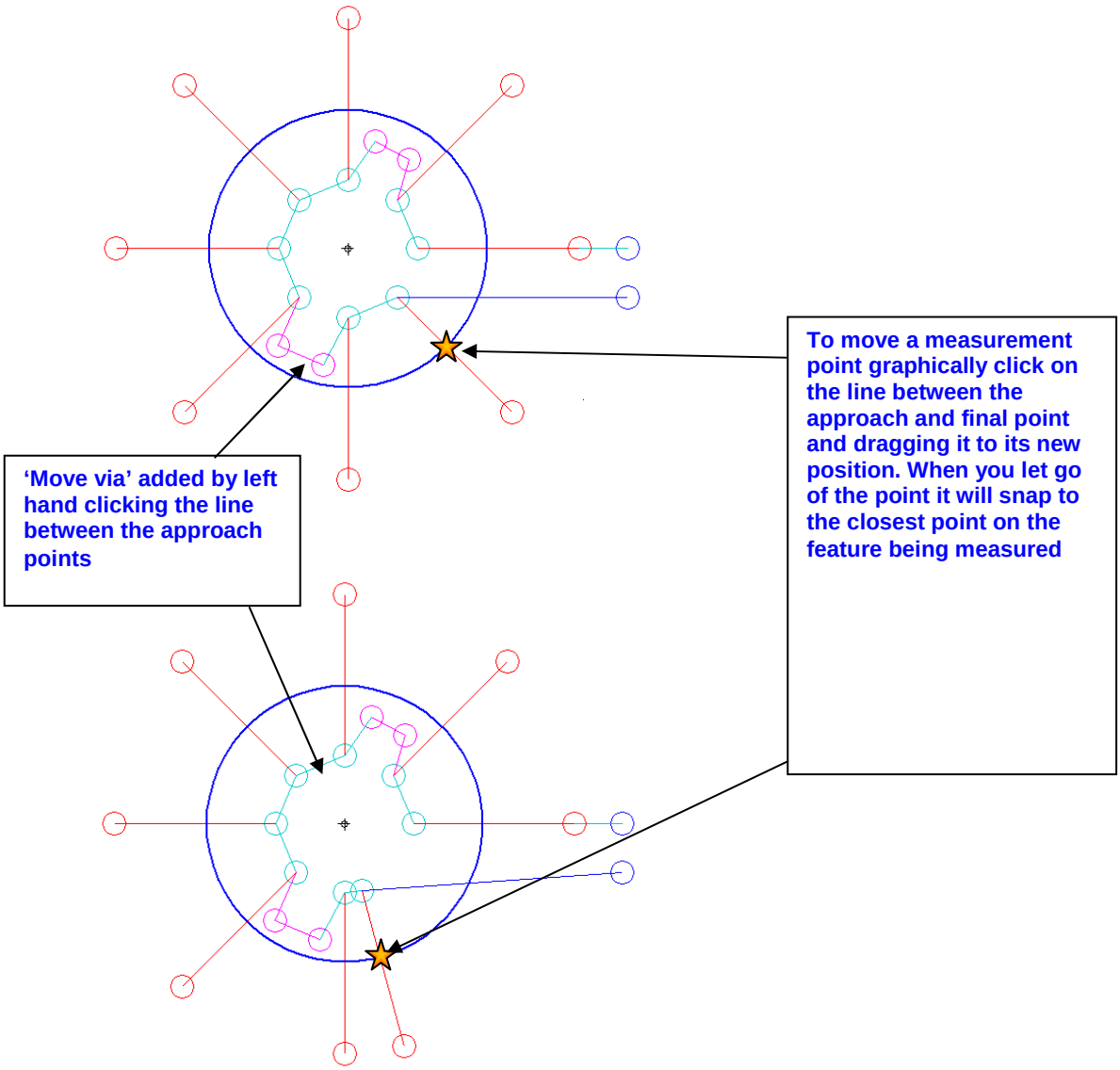
To turn the points off simply click on the button again.

38.1 Moving the measurement points

www.EngineeringBooksPdf.com

You can actually move the measurement points in a unit graphically by simply clicking on the desired point and dragging it to its new position. When you let go of the point it will snap to the closest point on the feature being measured.

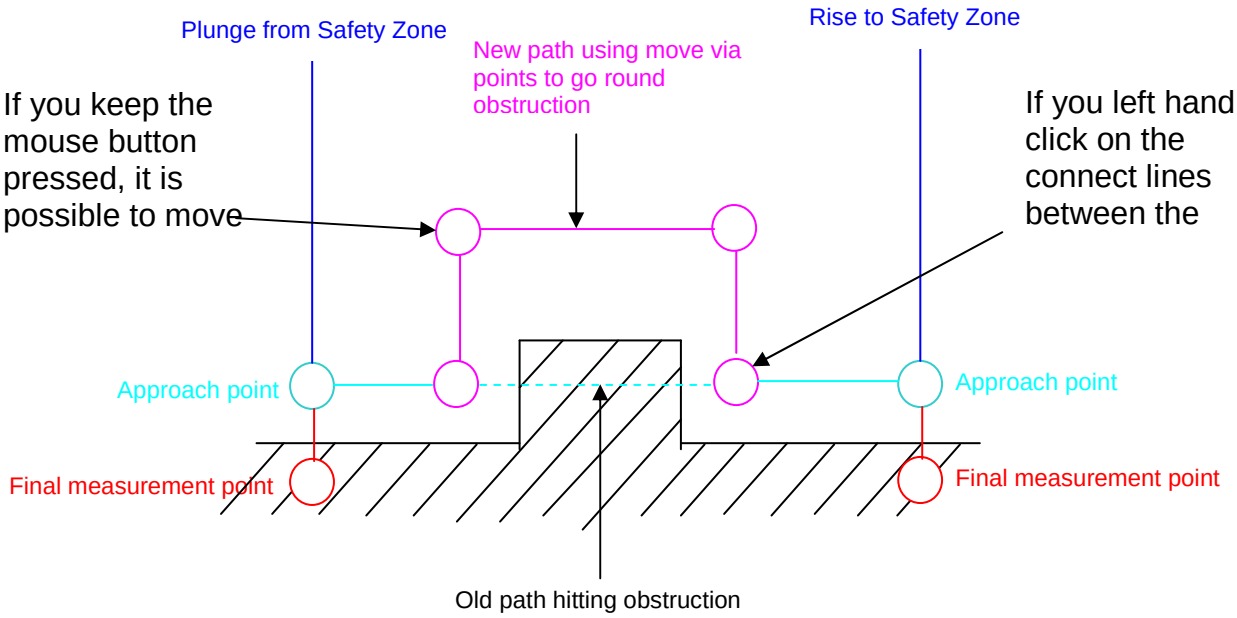
When you come to run the program automatically the software will re-prompt for the calculation of a safe volume and approaches.



www.EngineeringBooksPdf.com

38.2 Adding 'Move Via' graphically

You can add 'move via' points graphically by left hand clicking on the connect lines between the measured points this will add extra 'Move Via' graphically (shown as pink points), if you keep the mouse button pressed, it is possible to move the 'move via' points around on the screen so clearing any obstruction.



www.EngineeringBooksPdf.com

Website www.aberlink.co.uk
Email sales@aberlink.co.uk
Tel +44 (0)1453 884461

www.EngineeringBooksPdf.com

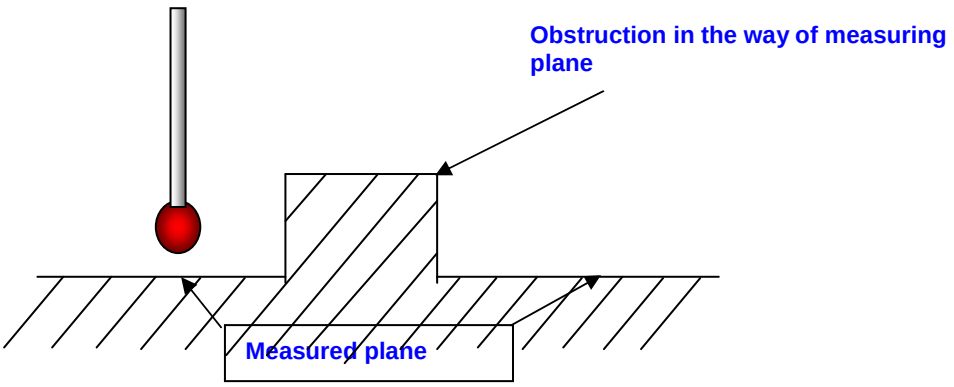
39.0 Inserting a 'Move Via' into a measuring routine

When the Aberlink 3D software creates a measuring routine it will measure the points on a feature in the most efficient way, ie if three points are measured on a plane the machine will measure the plane as follows:

1. It will rapid to the edge of the safe volume above the first point. [plunge position]
2. It will rapid to the pre-travel distance above point 1 (typically 2mm) [Approach 1]
3. It will probe point 1 and come back up to the pre-travel distance above the plane. [Final 1 represents the over travel distance past the point]
4. It will then rapid at the pre-travel height above the plane to a position above point 2. [Approach 2]
5. It will probe point 2 and come back up to the pre-travel distance above the plane. [Final 2 represents the over travel distance past the point]
6. It will then rapid at the pre-travel height above the plane to a position above point 3. [Approach 3]
7. It will probe point 3 and come back up to the pre-travel distance above the plane. [Final 3 represents the over travel distance past the point]
8. It will then rapid to the edge of the safe volume above point 3 (rise position). [Rise Position]

In other words when measuring the points on the plane the probe never moves more than the pre-travel distance above the plane.

So what happens if there is an obstruction in the middle of the plane between the points that the probe will crash into?



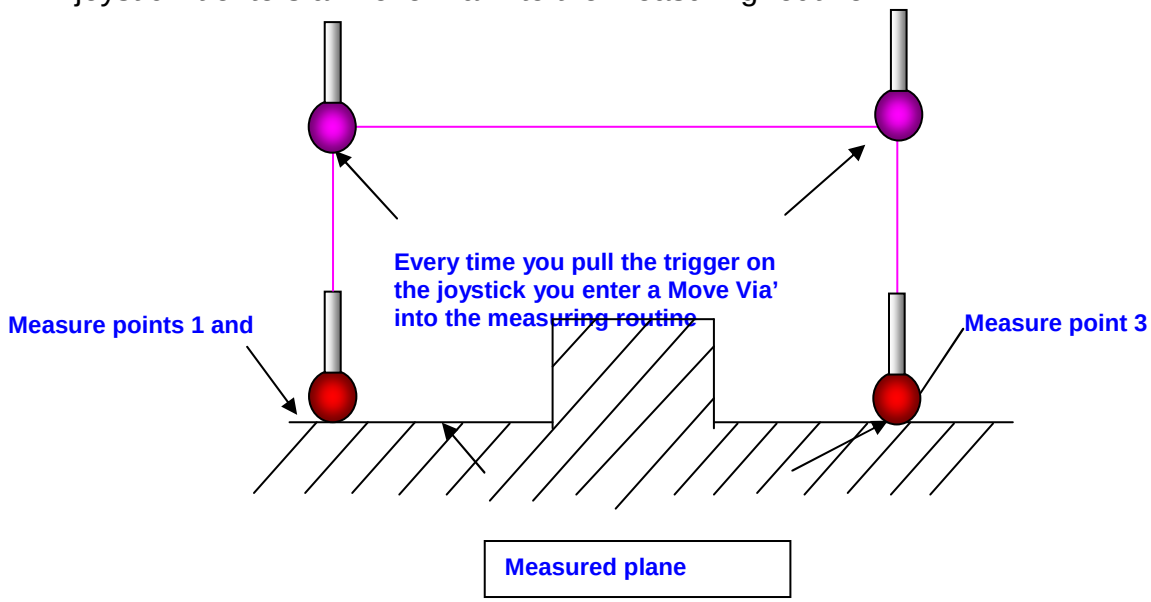
The answer is that you will have to insert a 'Move Via' into the measurement cycle for the plane. Say for instance that points 1 and 2 are on the left hand side of the obstruction and point 3 is on the right hand side. As the software would create the program the probe would crash into the obstruction after the first 2 points. We must therefore insert a 'Move Via' between the 2nd point [Final 2] and the move to above the 3rd point [Approach 3] that will take the probe up and over the obstruction.

www.EngineeringBooksPdf.com

There are three methods in which this can be done:

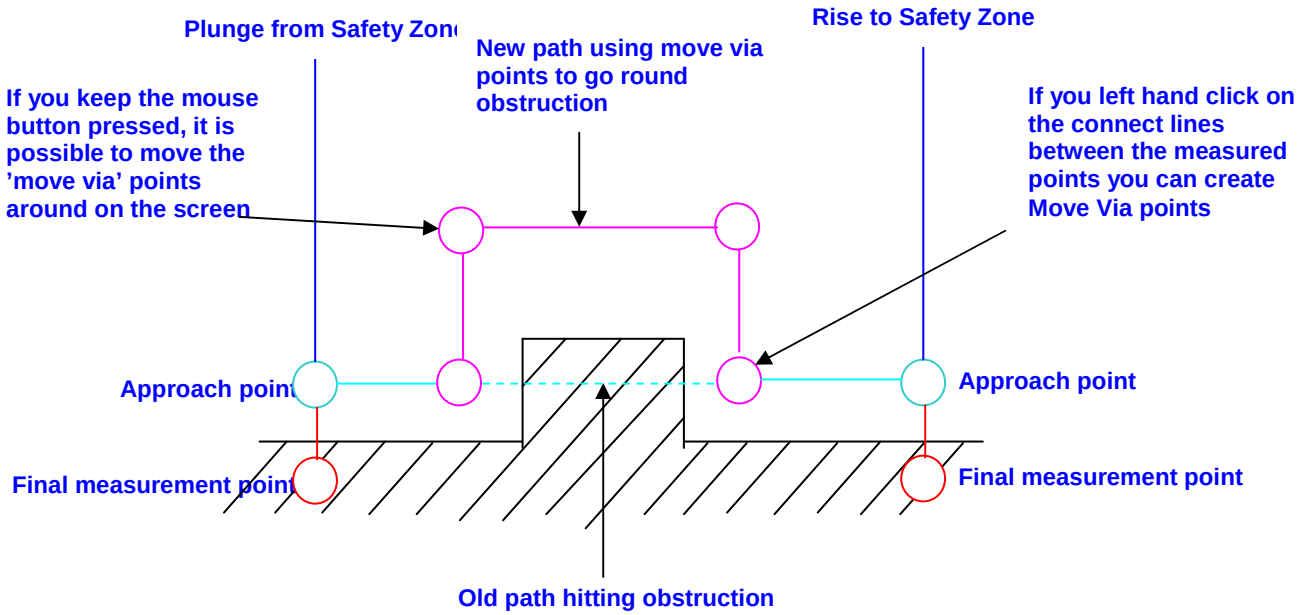
39.1 The Joystick Method

Using the joystick trigger button it is possible to enter the machines current position as a 'Move Via' command. Every time you press the trigger on the joystick it enters a 'Move Via' into the measuring routine.



39.2 On the Screen Method

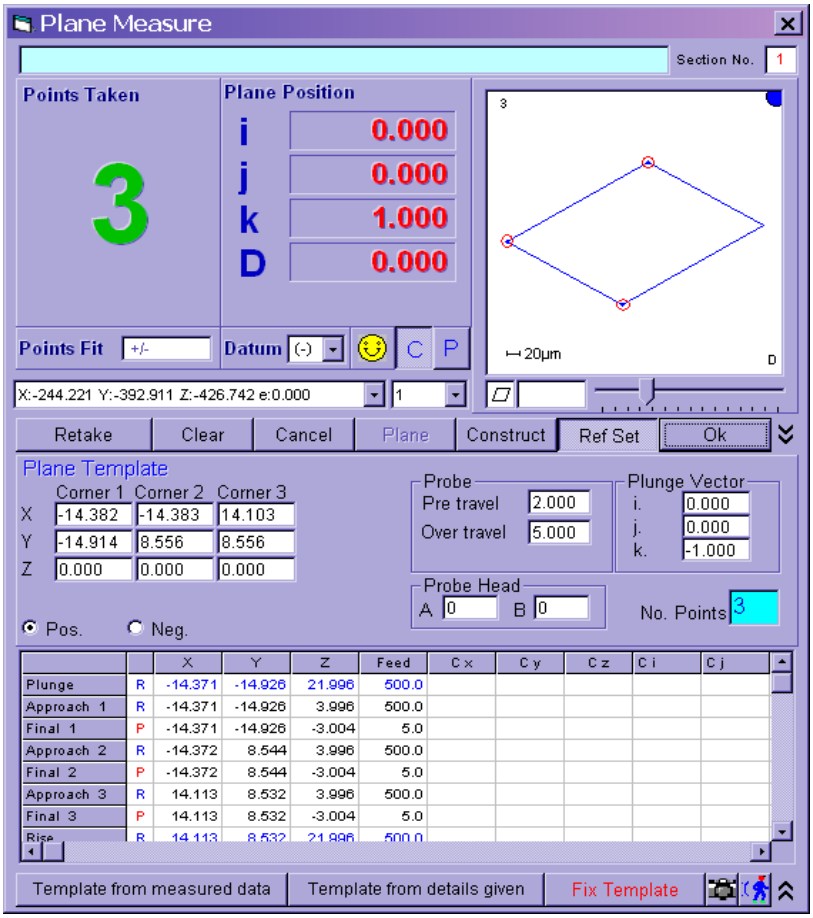
If you press this button you can see the path the probe takes along with the plunge, rise positions, pre-travel and measurement points graphically on the screen. You can edit these paths by left hand clicking on the connect lines between the measured points this will add extra 'Move Via' graphically (shown as pink points), if you keep the mouse button pressed, it is possible to move the 'move via' points around on the screen so clearing any obstruction.



www.EngineeringBooksPdf.com

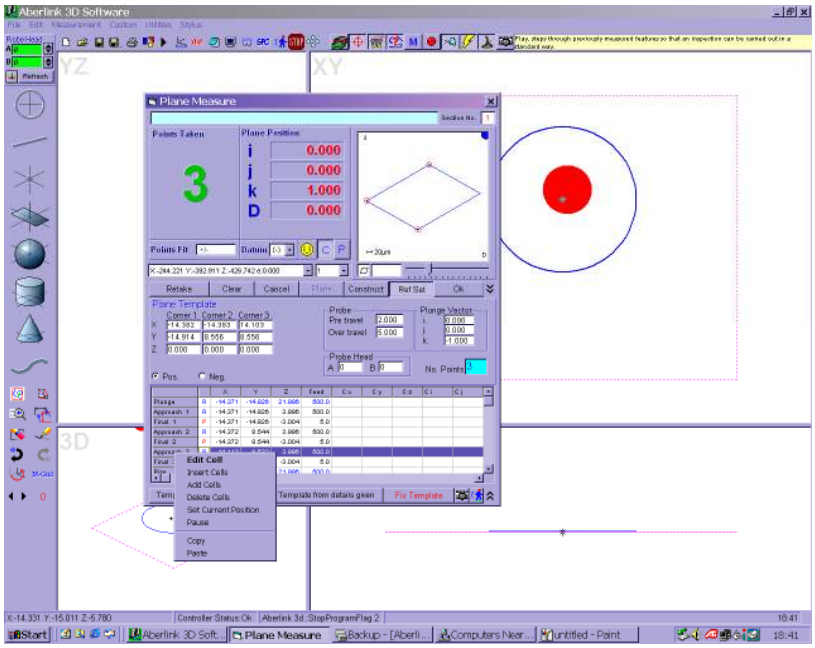
39.3 Edit Measurement Grid Method

You can type the X, Y, Z information, speed and probe head position directly into directly into the measurement grid.



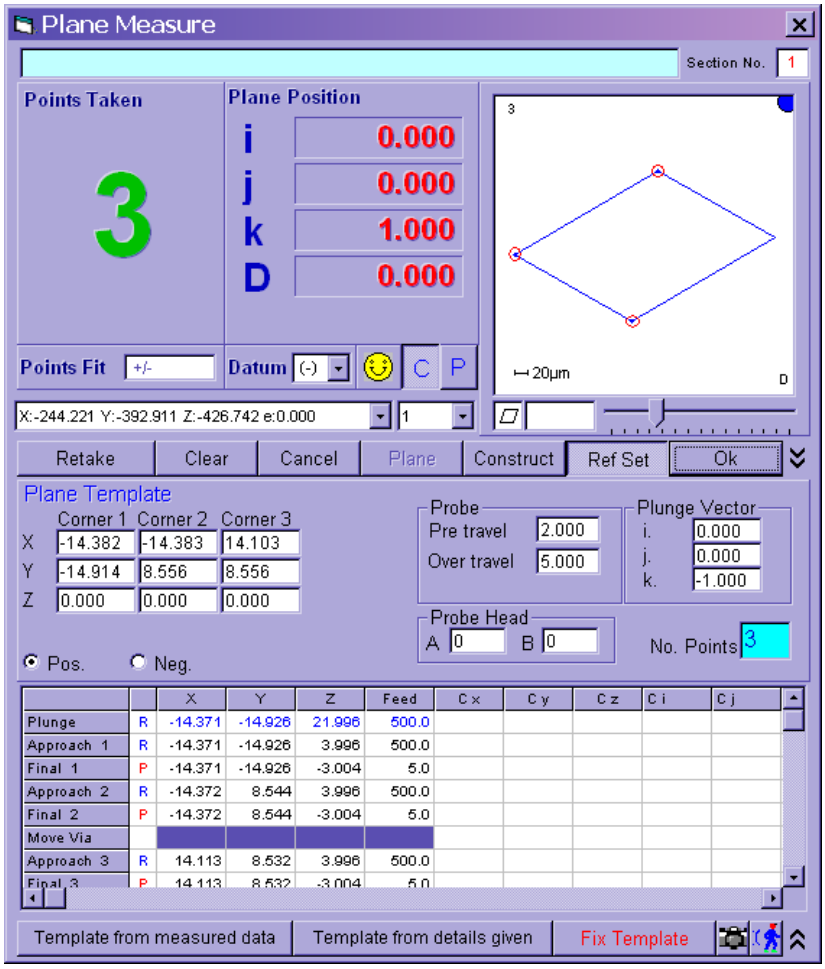
1. To enter a 'move via' between points 2 and 3 you need to left hand click on the 'Approach 3' box to select that row of the grid. The row will turn dark blue.

2. Now right click on the 'Approach 3' box to bring up the following menu:

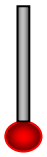


3. Select 'Insert Cells' and a blank move via line will be added into the programme:

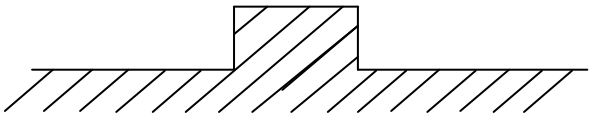
www.EngineeringBooksPdf.com



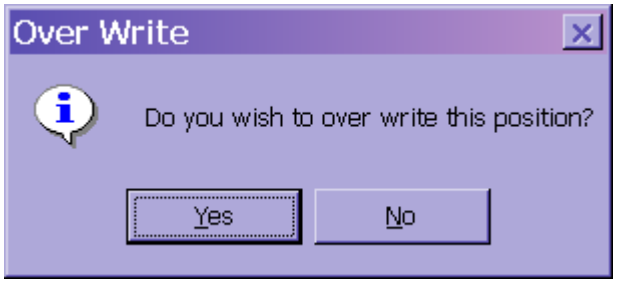
4. The X, Y, Z information and speed and probe head position could all be typed directly into this new row, if desired. It is much easier, however, to simply drive the probe to a suitable safe position above the obstruction and then click the trigger of the joystick to automatically enter this position into the Move Via line:



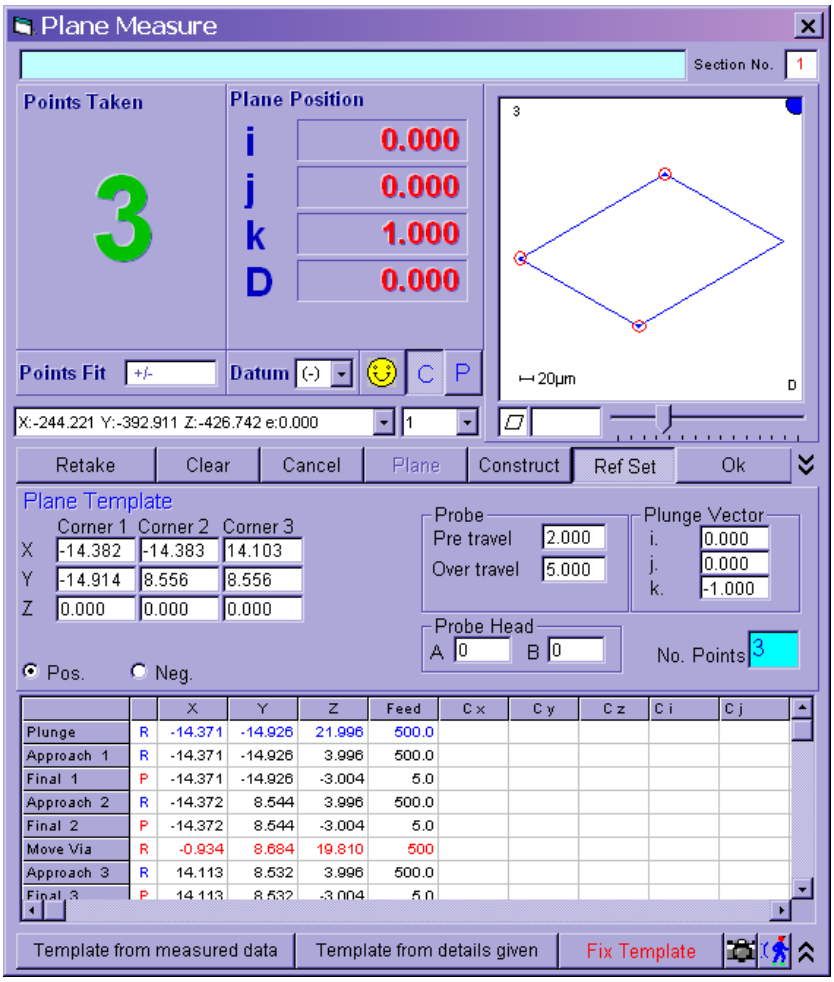
Click the trigger of the joystick when the probe is roughly in this position



When following prompt appears you can click either ‘yes’ or ‘no’. If you click ‘Yes’ the X, Y, Z position of the joystick will be automatically entered into the Move Via line. If you click ‘No’ the procedure will be aborted.



www.EngineeringBooksPdf.com



The ‘Move Via’ position will now be added. All you need to do is now click the ‘OK’ button.

DO NOT click ‘Template from measured Points’ or ‘Template from Details Given’ as these buttons will cause the probe path to be recalculated.

The program will now move as follows: It will take points 1 and 2 staying at the pre-travel height above the plane. It will then move to the inserted position above the obstruction before moving down to the pre-travel distance above point 3, and then take point 3 before going up to the edge of the safe volume.

www.EngineeringBooksPdf.com

40.0 THE STORE POINTS FUNCTION

The Store Points function allows the software to record measurement points taken, even when it is not expecting any.

To put the software into this mode, click on the Store Points button.

This button is a latching type, which when on, will become highlighted. To switch the Store Points function off again you must click on the button for a second time, and the button will return to its normal grey colour.

With the Store Points function switched on, the software will now memorise every measurement point taken. Now if you measure a feature before having clicked on the particular measurement button, you have not wasted your time. Each point will now make the usual measurement point sound (rather than than 'No Form Open sound). After taking the measurement points, simply click on the required measurement button, and the software will fit the stored measurement points to that feature.

Be careful however, because sometimes the probe may be triggered when you are not taking a genuine measurement point, for instance when you are changing the probe stylus. The software will still record these points, and the next time a measurement window is opened, it will try to fit these points to the feature. The result is likely to be some massive errors, and so if this does happen, you need to erase any erroneous points, or simply clear all the stored points before you commence your measurement. See Chapter 15 – Erasing Data Points.

www.EngineeringBooksPdf.com

41.0 THE FEATURE PREDICT FUNCTION

The feature predict function is a time saving function, aimed at cutting down the amount that the user has to interface with the PC. In this mode the software will predict the feature required to fit the measurement points taken.

This function will work for 2 dimensional features, such as lines, circles and planes, but cannot be used for either one dimensional points or 3 dimensional features such as cylinders, cones or spheres.

To put the software into this mode, click on the Feature Predict button.

This button is a latching type, which when on, will become highlighted. It will also automatically bring on the Store Points function, which will also become highlighted. To switch the Feature Predict function off again you must click on the button for a second time, and the button will return to its normal grey colour. Note that the Store Points function will also have to be turned off separately.

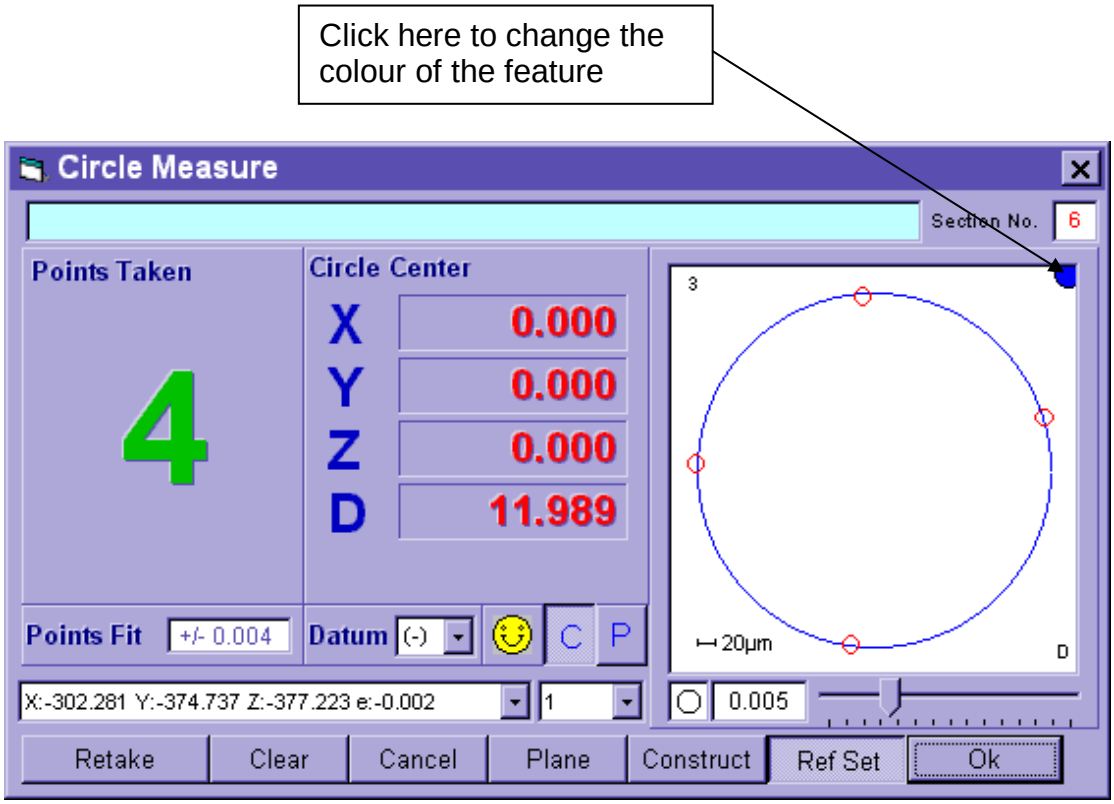
With the Feature Predict function switched on, you can immediately start to take the measurement points on a feature that you are measuring, without clicking on the measurement button first.

If the first 3 measurement points taken can be fitted to a line, a Line Measurement window will automatically appear on the screen displaying the points taken.

www.EngineeringBooksPdf.com

42.0 CHANGING THE COLOUR OF A FEATURE

It is now possible to change the colours that features appear on the screen. In the top right hand corner of the graphic representation within any measurement window there will be a dot of the same colour that the feature will appear. To change the colour click on this dot.



This will bring up a colour palette on the screen:

www.EngineeringBooksPdf.com

42.0 Display Measured details or template details or

When a program is first made the software will create a template for each feature, that will be used for each subsequent measurement. Usually this information will be taken from each feature on the first time that is was measured, however, it is also possible to download a program template from a DXF file. This information can be seen in the template area of each measurement window.

When the button shows ‘M’ the Main Screen will show measured features drawn in blue.

If you click on the ‘M’ the button will change to ‘T’ and the template for the inspection will now be shown in grey.

Click on the ‘T’ again to change it back to ‘M’.

This function has lttle significance on a manual machine .

www.EngineeringBooksPdf.com

43.0 INPUTTING AND OUTPUTTING DXF FILES

Introduction

The DXF function outputs the current drawing in DXF file format and imports hole position / template data from a DXF file.

Fig. 29.0-1 The DXF File button

When this button is pressed the following form is displayed;

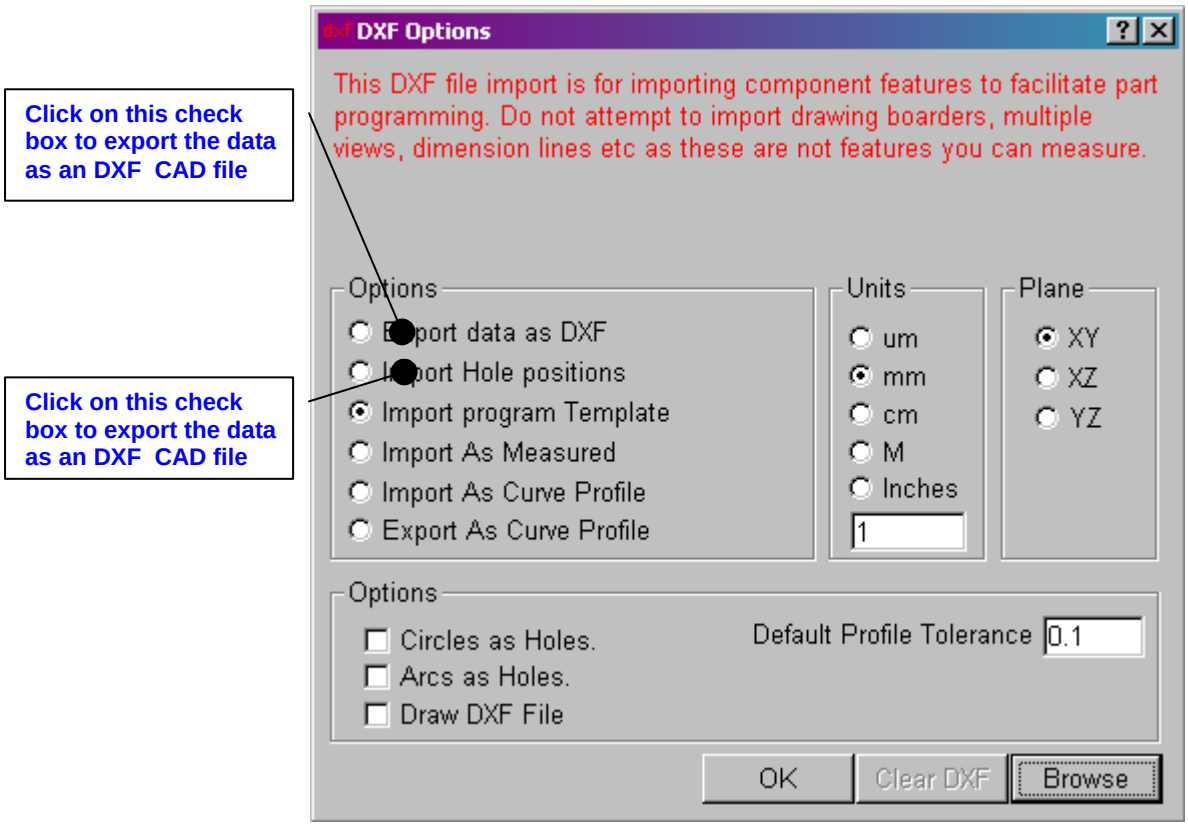


Fig. 29.0-2 The DXF Options window

This option will allow the measured data to be exported as a DXF file. The geometry of the measured features can then be imported into a CAD package, for example for reverse engineering purposes.

Select the ‘Export Data as DXF’ option, and then use the ‘Browse’ button to save the DXF file where desired.

Note that this will export the measured features only. The dimensions, which may have been called up are not compatible with a DXF file format.

www.EngineeringBooksPdf.com

43.1 Import Hole Positions

The Point Unit Details report will compile a tabulated report of the nominal and actual measured XY centres of all circles arcs and points in an inspection. The nominal values for the centres can either be entered by typing in the values within the template area for each unit, or alternatively by importing the hole positions using a DXF File.

Select the 'Import Hole Positions' option and browse to the relevant DXF File. The software will now match each measured hole to its corresponding position in the DXF file, so long as it is within the tolerance set under the 'DXF Hole Found Tolerance' setting under the 'Units' tab in the Machine Set Up window. The holes within this tolerance will temporarily change colour on the screen, and their nominal XY centre positions will automatically be updated.

Note that the reference point (0,0 position) and alignment for the measurement must match that of the DXF file.

43.2 Import Program Template

This function can be used to import a DXF file, which can then be run as a program using the Play function. The features to be measured will appear as light grey on the Main Screen. When the program is run the feature to be measured will turn green, and its measurement window will automatically open.

The advantage of this is minimal on a manual machine, such as Project X.

43.3 Import As Measured

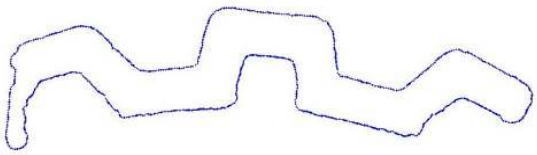
This function exactly the same as above. A DXF file can be imported and run as a program. However, the only difference with this option is that the features to be measured will appear as dark blue (measured features) rather than light grey (template).

43.4 Import As Curve Profile

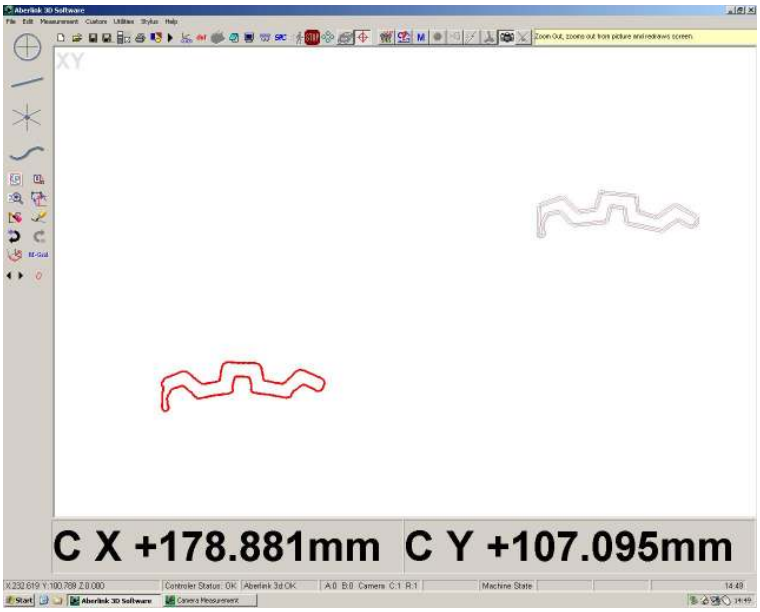
If the component shape has been measured as a Curve profile, it is possible to import the DXF file for the shape, and perform a best-fit optimisation to get the measured points to fit the DXF file.

Firstly measure the component as a Curve. This will appear as a series of blue dots in the Main Screen:

www.EngineeringBooksPdf.com

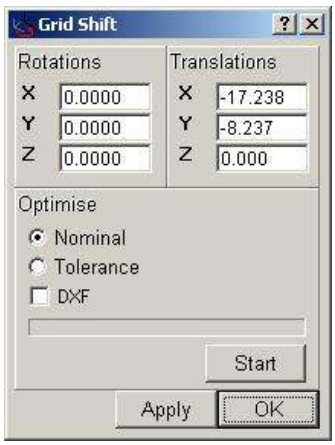


Select ‘Import As Curve Profile’ in the DXF window. Also type in the default profile tolerance required, and then browse to the DXF file. When you OK the DXF window the DXF file will be drawn in the Main Screen, but will not necessarily be in the same position as measured component, so it may be necessary to zoom out in order to see it:



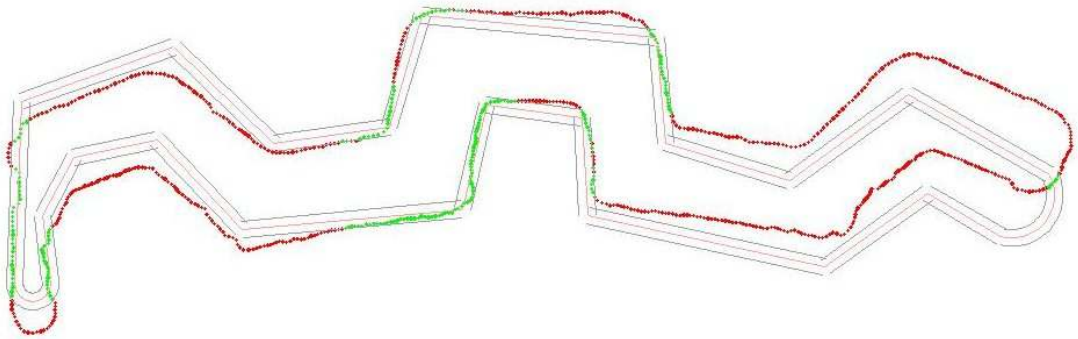
The nominal DXF file is shown as a red line, with the default tolerance shown as grey lines on either side.

Now open the Grid Shift window as follows:



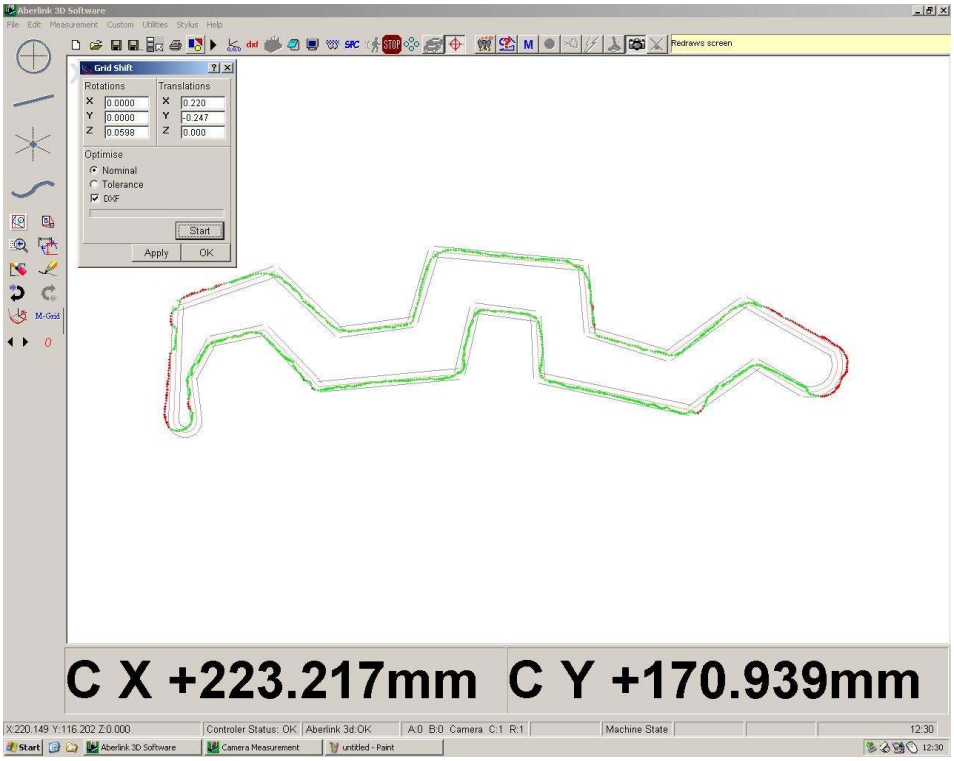
www.EngineeringBooksPdf.com

Whilst this window is open, the measured data points can be dragged using the right mouse button so that they are approximately over the top of the imported DXF file:



You will see that where the points are within the specified tolerance band they are shown in green, and where they are outside they are shown in red.

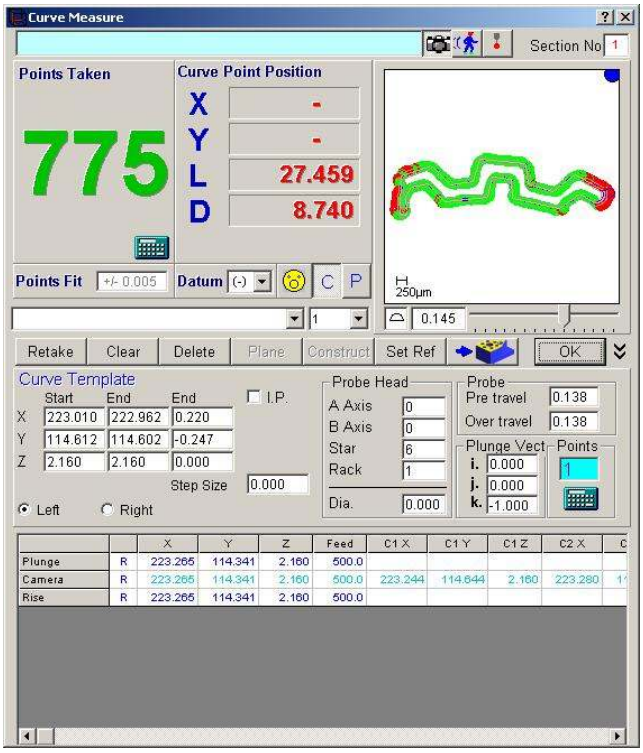
Now to perform a best fit, tick the DXF box under the ‘Optimise’ option and then click start. The software will now take a few seconds to calculate the rotation and translations required to fit the measured data to the DXF file. When it is finished the measured points will snap to the DXF file. Click OK and the software will return to the Main Screen:



At a glance the areas outside of the tolerance can be seen as they are shown in red.

Right clicking on the representation of the curve (or using the Feature Select buttons) will re-open the Curve Measure window:

www.EngineeringBooksPdf.com



Right click in the graphic area and choose ‘expand’ to make the graphical representation fill the whole screen:

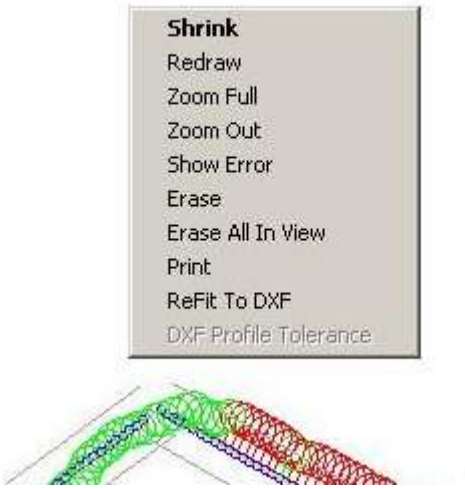
The measurement point shown in blue denotes the biggest error.

This value will be shown as the Profile of a Surface value in the bottom, left hand corner of the window.

www.EngineeringBooksPdf.com

The scale can be modified by using the slider bar, which is located next to the Profile of a Surface box. A scale indicator line is shown above the slider bar.

Right clicking in the window will bring up the following menu:



Shrink - will return to the full Curve Measure window.

Redraw - will refresh the image.

Zoom Full - will zoom to fit the whole of the Curve to fit the window.

Zoom Out - will zoom out by a fixed percentage.

Show Error - will bring up a box showing the error of any point right clicked on.

Erase - will erase any point right clicked on.

Erase All In View - will allow a box to be dragged over multiple points to be erased.

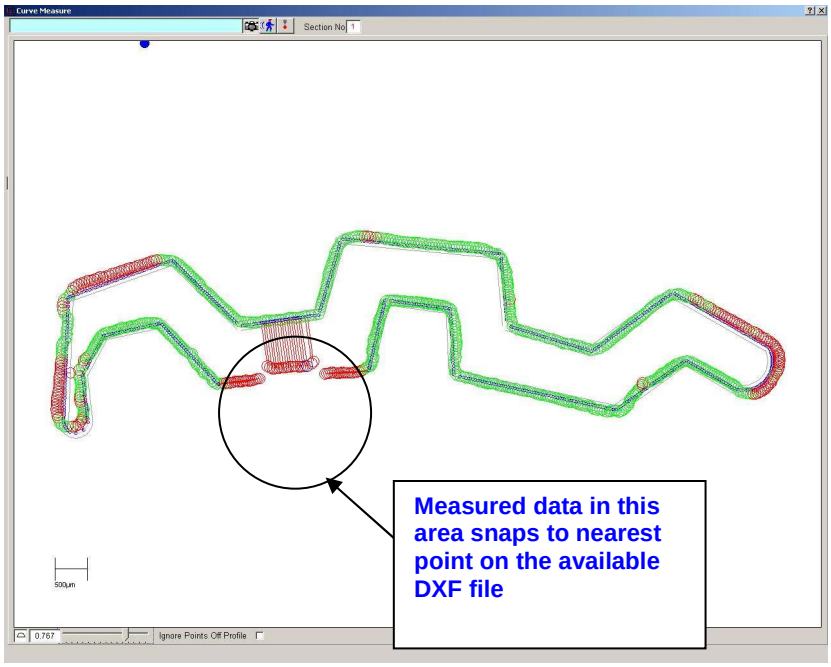
Print - will allow this representation to be printed as a Feature Profile report.

ReFit to DXF - will allow the best fit routine to be repeated.

DXF Profile Tolerance - will allow the default tolerance that was imported with the DXF file to be changed for any feature within the DXF file. Right click on that feature and select this option.

If the DXF file happens to contain gaps for any reason, this can produce errors because the error for the measured data within the gap(s) will be reported to the closest point on the DXF file, as follows:

www.EngineeringBooksPdf.com



In this situation simply tick the 'Ignore Points Off Profile' box, which will now produce a representation as follows:

www.EngineeringBooksPdf.com

43.4 Export As Curve Profile

When a DXF file is imported as a Curve Profile, the default tolerance will be applied to all features within the DXF file. It is possible to change the tolerance for each individual feature of the DXF file by right clicking on it and selecting the ‘DXF Profile Tolerance’ option, as described above.

If you wish to save the DXF file again complete with the modified tolerances, then the Export As Curve Profile option can be used.

www.EngineeringBooksPdf.com

44.0 THE PLAY FUNCTION

When inspecting multiple components, you may wish to measure them in a similar manner to each other. In this situation the 'Play' function will prove to be useful.

When you have finished the inspection of the first component of a batch, or have called up a previously saved inspection, click on the 'Play' button

Fig. 44.0-1 The Play button

The play function will now take you through the inspection of the part in the same order as previously measured, by highlighting in green each of the features previously measured in turn.

For instance, consider an inspection that consisted of a plane, a line and a circle measured in that order, with a single dimension giving the distance between the line and the circle.

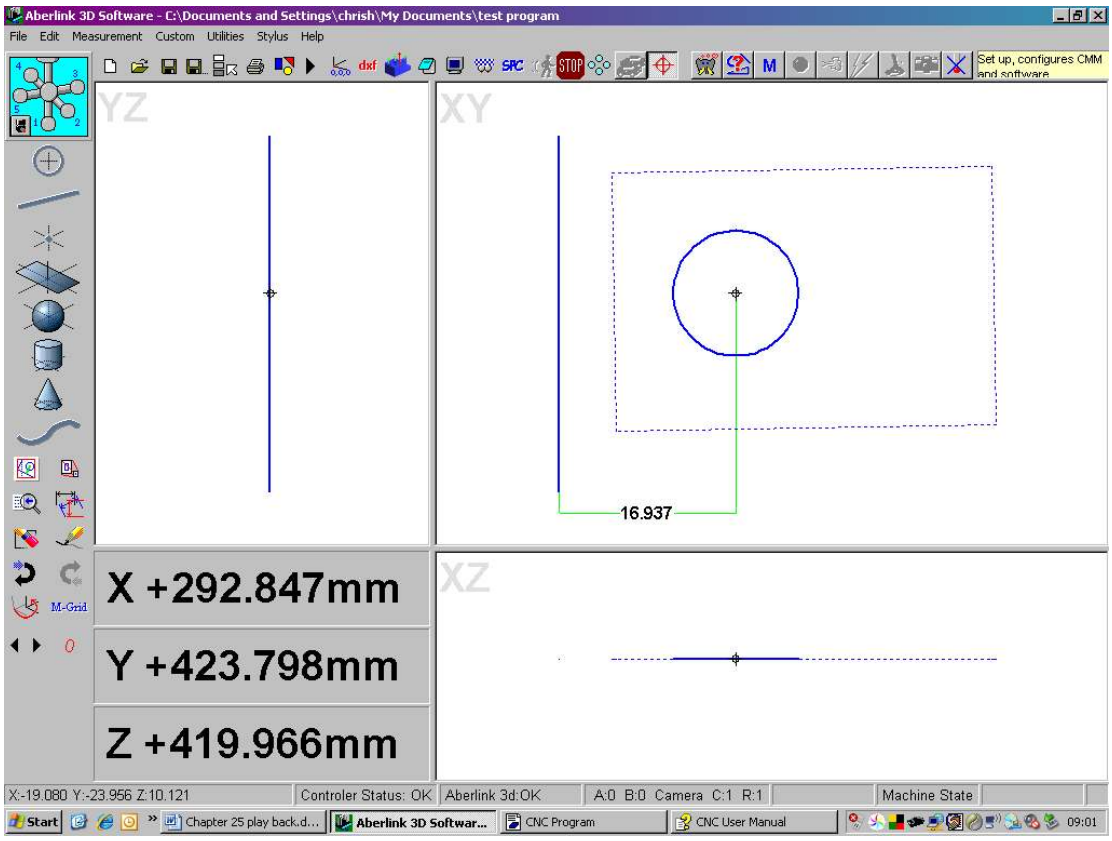


Fig. 44.0-2 An example of a simple inspection

Now to measure a similar component in the same manner, click on the Play button, this will open the program window shown below

www.EngineeringBooksPdf.com

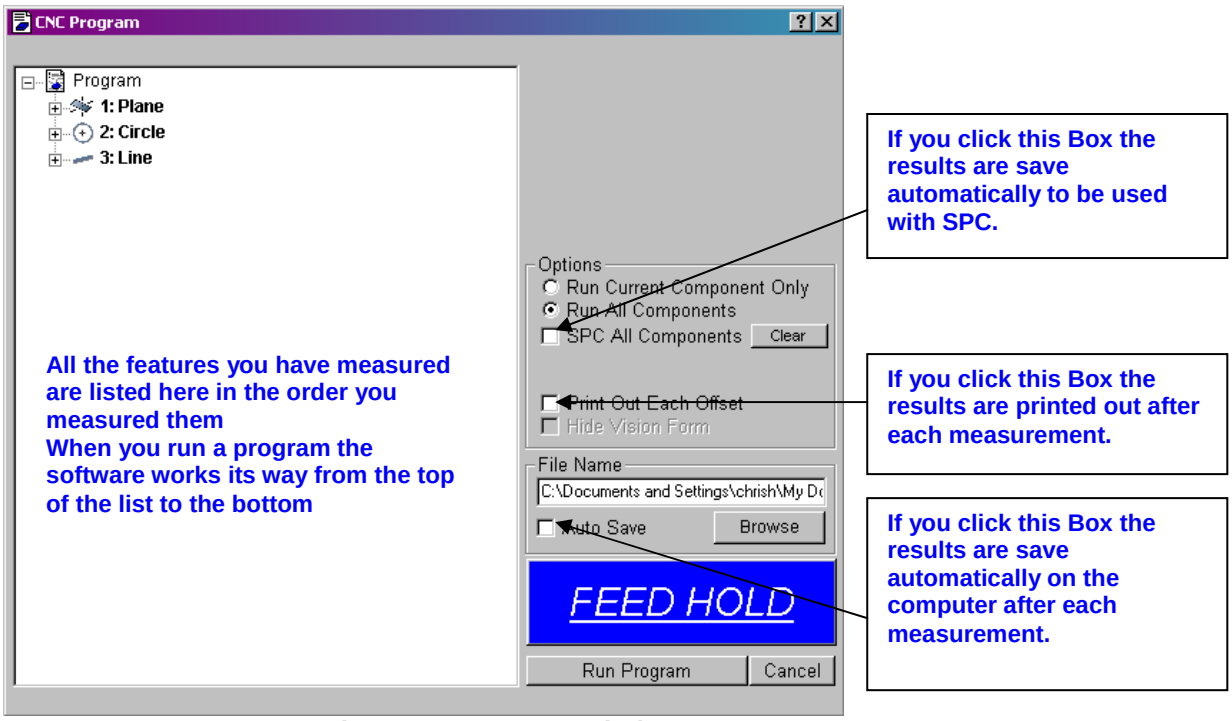


Fig. 44.0-3 Program Window

To run the program press the 'Run Program' button this will close the program window and open the 'Program Notes window' as shown below

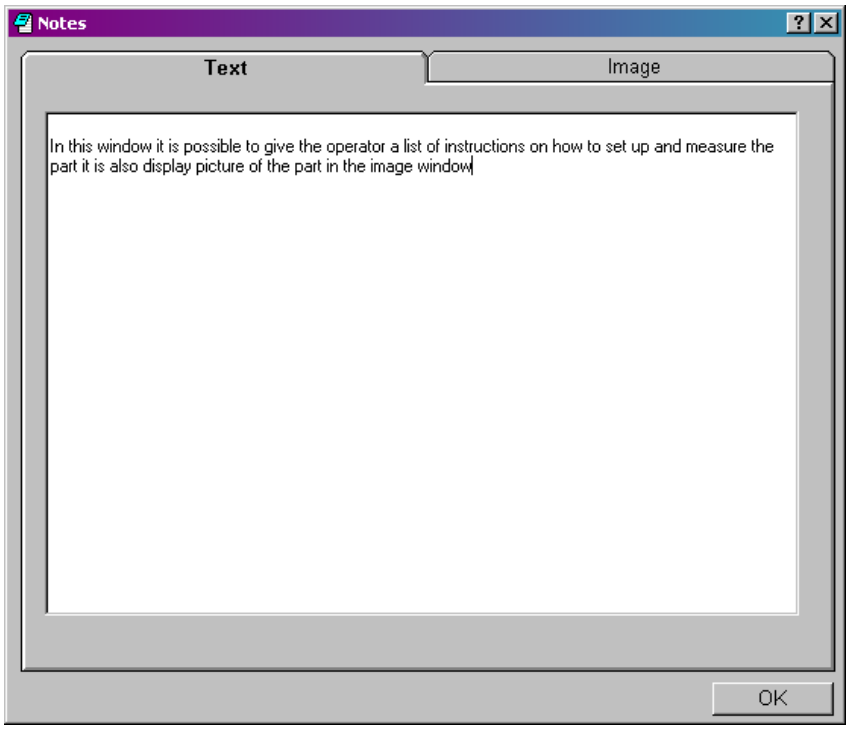


Fig. 44.0-4- Program Notes Window

In this window it is possible to give the operator a list of instructions on how to set up and measure the part it is also possible to display picture of the part in the image window.

www.EngineeringBooksPdf.com

After you have read the instruction click the 'OK' button this will close the notes window and the plane that was originally measured (*plane No1 in the list shown in the program window*) will become highlighted in green, and the Plane Measure window will automatically open.

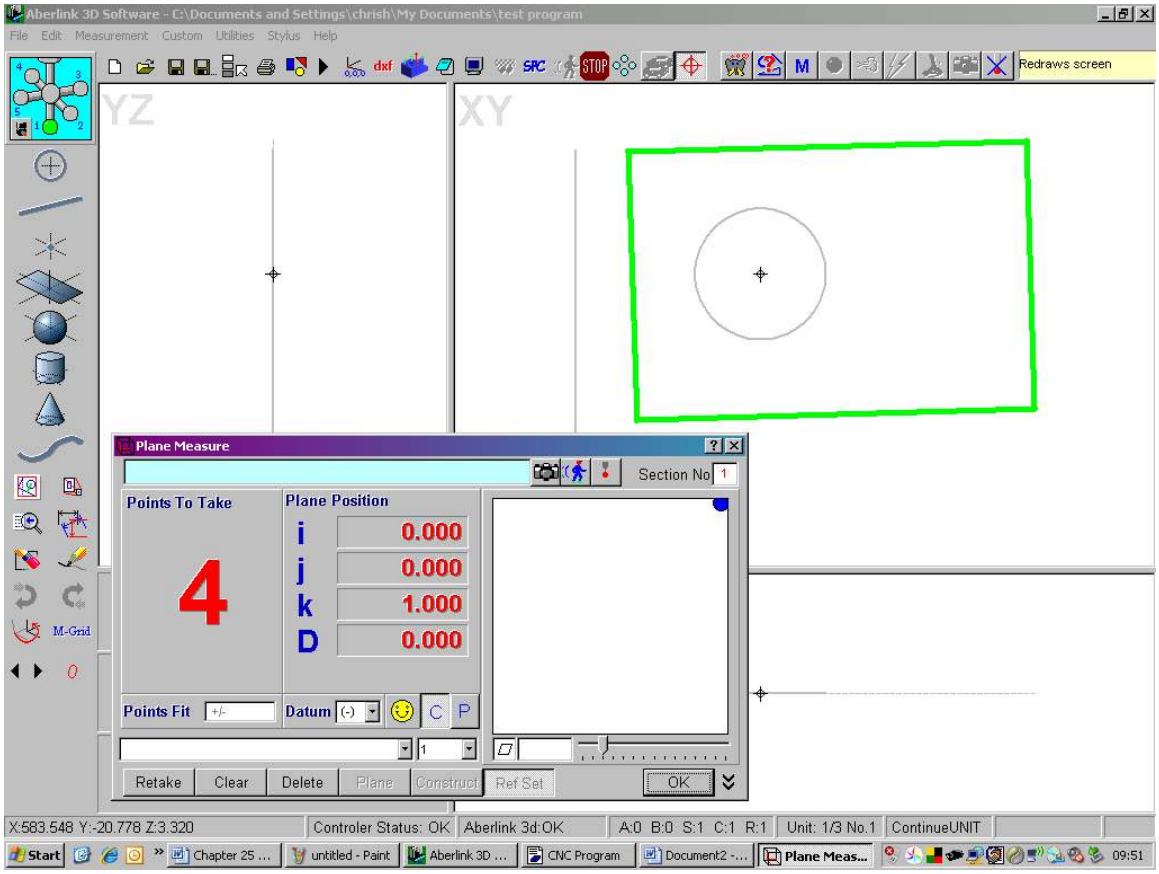


Fig. 44-5 Showing the first feature to be re-measured using the Play function

Now measure 4 points on the plane on the second component as you take the points the 'Points to take' window will count down to zero when the count reach's
Zero you have finished measuring the plane.

Note:- At any time before you taking the last point you can retake or clear points

www.EngineeringBooksPdf.com

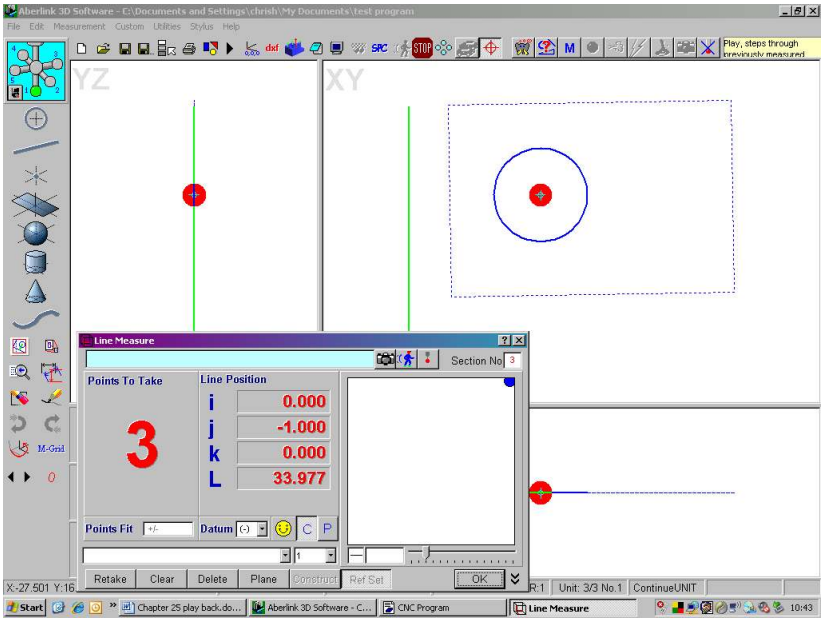


Fig.44.0-6 Showing the second feature to be re-measured using the Play function

On completion of the plane measurement, the second feature, in this case the line, will become highlighted in green, and the Line Measure window will automatically open.

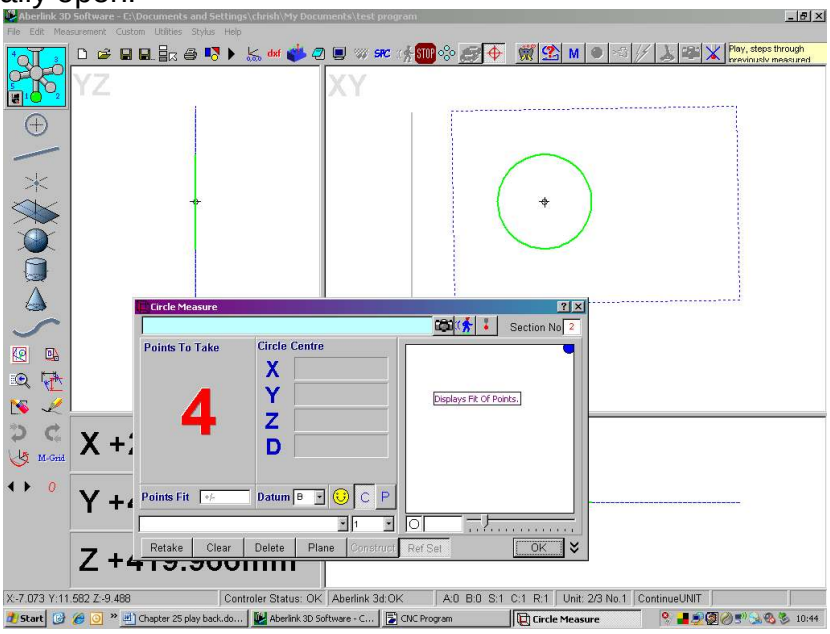


Fig. 44.0-7 Showing the third feature to be re-measured using the Play function

After measuring the all points in the line the 'line window' will close and the 3rd feature in this case the circle, will become highlighted in green, and the circle Measure window will automatically open.
Repeat the process for all the features listed in the program window, After you have measured the last feature in the list the 'Inspection Results' window will be displayed telling you if the part has pass or failed inspection.

www.EngineeringBooksPdf.com

If the part has failed inspection you will get a list of all the feature that have failed in the 'Inspection Results' window which you can save or print.

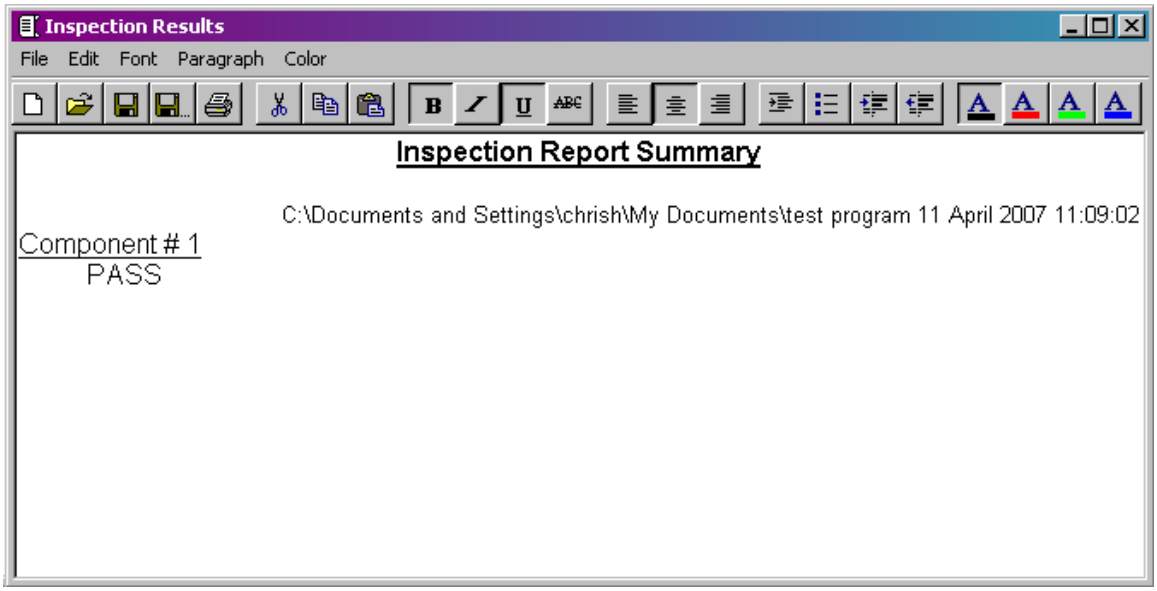


Fig. 44.0-8 Inspection Report Summary Window

To repeat the inspection again for a third component, you must click on the Play button again.

44.0.1 Useful Hints

1. To abort the Play function in the middle of an inspection, simply click on the Play button again.

2. If you cancel a measurement window, this will not only abort the Play function, but will also erase the particular feature from the inspection.

3. Each Measurement window can contain text in the light blue bar at the top – see chapter 18.2 Labelling Specific Measurement Units. This may be useful for identifying similar or close units.

4. The Counting Down feature may also be useful during the Play function, as the window will then automatically close when the required number of data points have been taken. The reference status will be set as for the previous feature. (this is set as standard.)

5. When using the Play function, it will help to have the components situated at approximately the same position on the table, in order to avoid the necessity to zoom in and out between previous and current inspections.

6. The Play function can also be used on inspections that have previously been saved. Hence for the first component of a regular batch, the previous

www.EngineeringBooksPdf.com

inspection can be recalled, and the play function used to obtain results using an identical inspection method.

7. If an inspection consists of constructed features as well as measured ones, these will be automatically re-calculated when the relevant measured features have been re-measured.

44.1 More information on the Program Tree

All the measurement units are shown in the program tree on the left hand side of the window. The reference features will be shown in bold.

More information about the units can be obtained by clicking on the plus sign adjacent to it. It is possible to look at the template for the unit and even the probe moves that will be made, although it is not possible modify the moves at this window. This must be done by recalling the unit from the main screen and modifying the probe moves grid within the measurement window.

You can modify the order of the program by clicking on a unit within the tree and dragging it to a new position. Care however, must be taken not to infringe the basic rules of the Aberlink 3D measuring software such as dragging a circle or a line in front of the Plane into which they are projected, for instance.

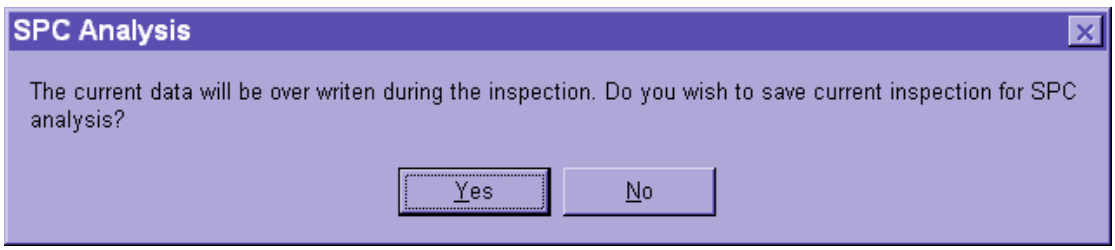
It is also recommended not to move the position of the reference features.

Finally you will discover that it is not possible to drag a unit to the very bottom of the tree. If you need to do this you will have to drag the unit as low down the tree as possible (to the penultimate position) and then drag the last unit above it. Also it is possible to run only part of a program, if desired. Right clicking on a unit on the tree will bring up the following menu:

44.2 Collecting SPC Batch Information

The only other useful option when using the Play function is the ‘SPC All Components’ option. The Aberlink 3D software will collect batch information as well as calculate some simple SPC information and plot both a histogram and a scatter graph for any given dimension.

In order to collect this information when running a batch of components you need to tick the ‘SPC All Components’ box on the Program Window.
Now when you click on ‘Run Program’ you will get the following message box:



www.EngineeringBooksPdf.com

If you wish to add the previous inspection to the batch information click on ‘Yes’, otherwise click ‘No’.

The batch information can be displayed within the Dimension Details window for each dimension - this is detailed in chapter – Showing SPC Batch Information.

www.EngineeringBooksPdf.com

45.0 INSPECTION NOTES

If you wish to save text or images with the inspection results, then click on the Inspection Notes button:

Fig. 45.0-1 The Inspection Notes button

This will bring up the Inspection Notes window:

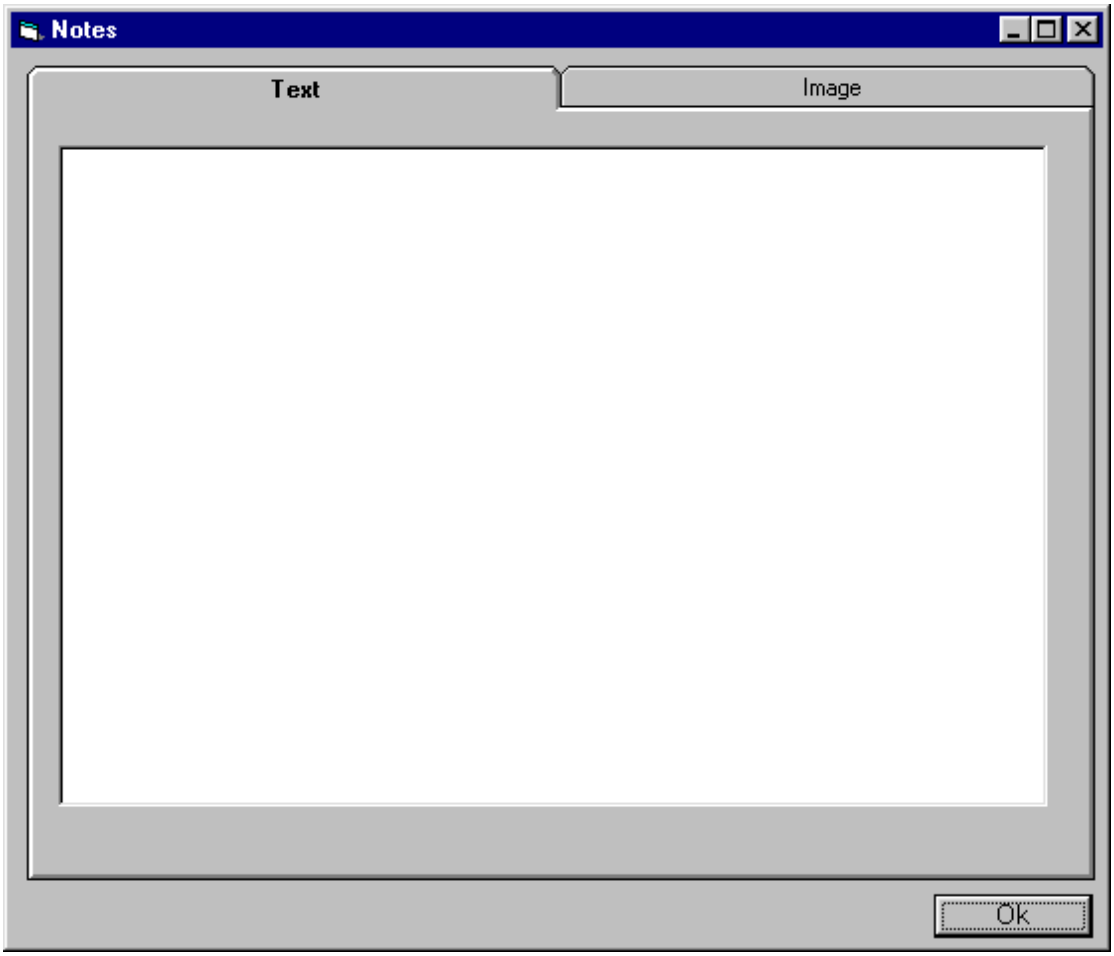


Fig. 45.0-2 The Inspection Notes window

45.1 Adding Text

The inspection window shows two tabs ‘Text’ and ‘Image’. The text area allows any inspection notes to be added e.g. comments on the number of points to take on a particular feature, or the type of stylus to use.

To add text click on the ‘Text’ tab. The background of the window should now be white. Click on the background area to bring the cursor into the window. Now simply add the required text using the keyboard.

www.EngineeringBooksPdf.com

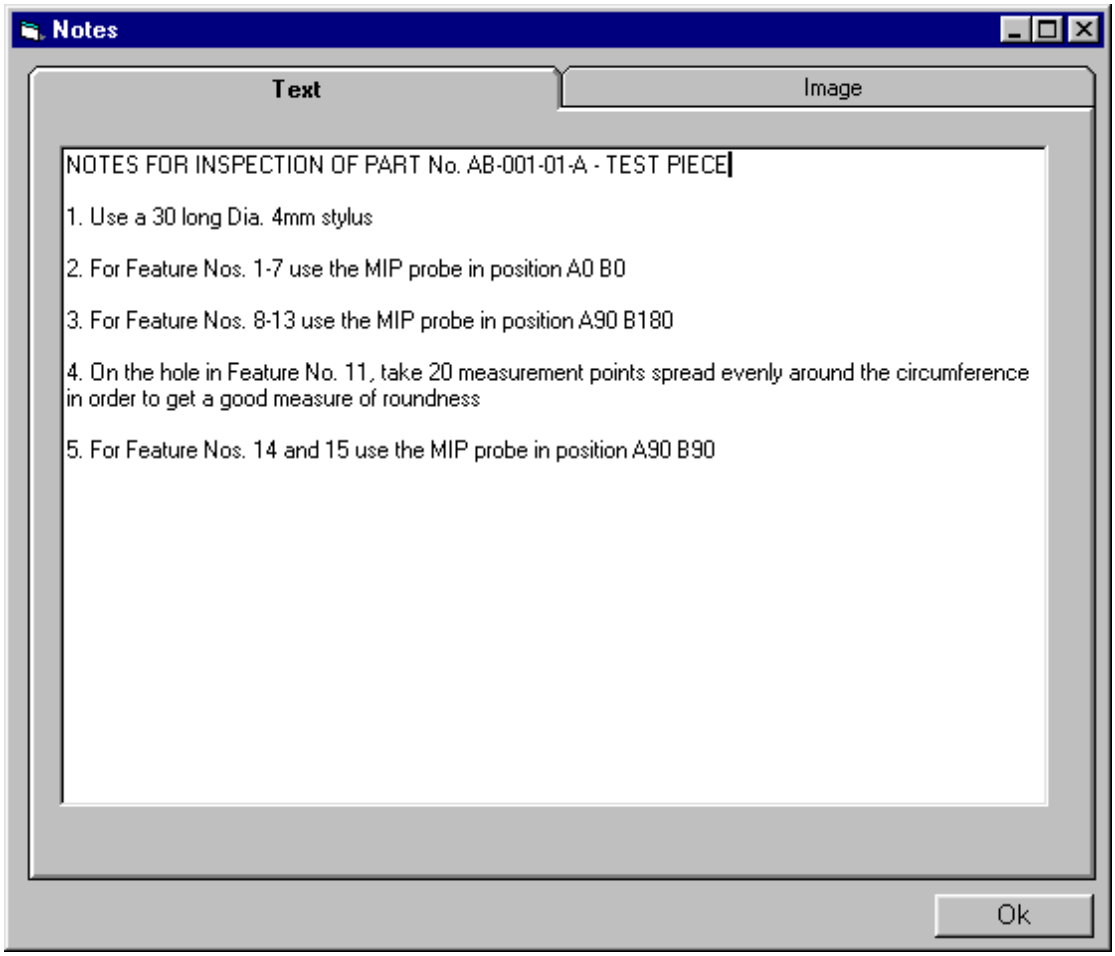


Fig. 45.1-1 Showing some saved inspection notes

45.2 Adding Images

The image area allows an image to be imported, to help identify similar components. To import an image click on the Image tab, and then the 'Load Image' button. A windows Open dialog box will appear to allow input or selection of an image file. Bitmap, metafile, enhanced metafiles, JPEG and GIF files can all be imported. When the inspection is saved, the software saves a pointer to the image not the image itself. This means that if the image is erased from the disk or moved into a different folder the inspection software will not be able to display the image.

www.EngineeringBooksPdf.com

46.0 DISPLAY LAYERS

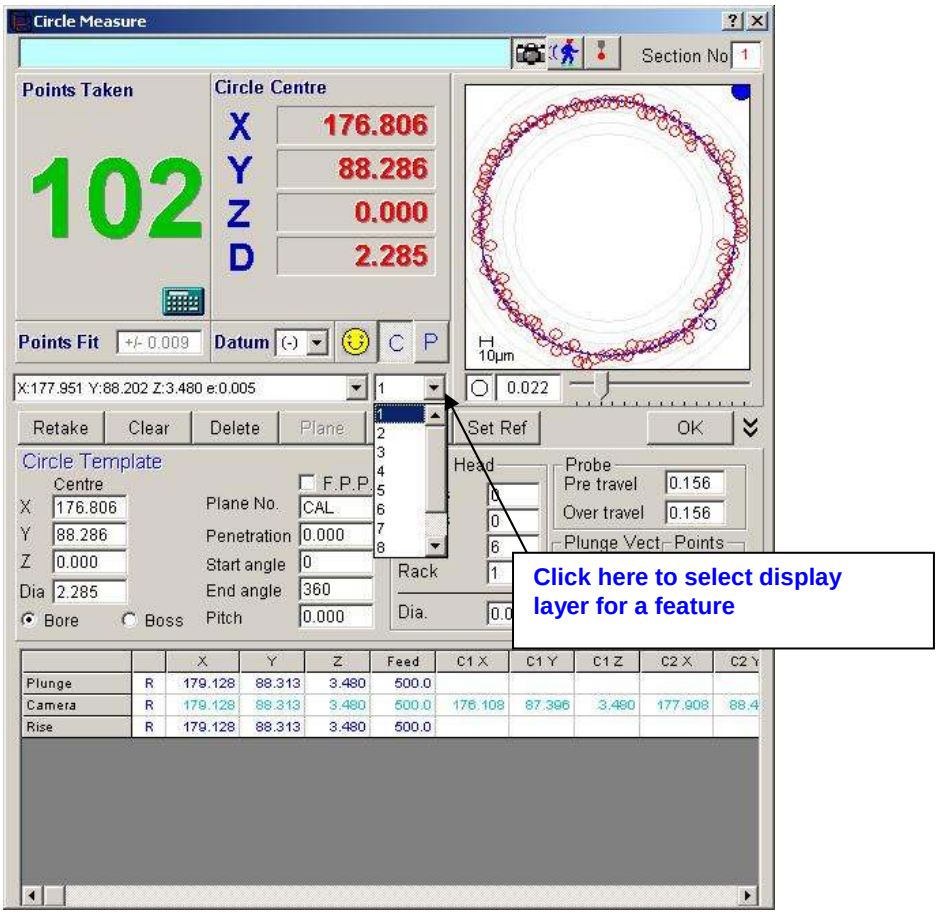
The main screen can occasionally become too cluttered to display information clearly. For this purpose the features and dimensions can be placed in up to 10 display layers, which may then be switched on and off to make the feature or dimension either visible or invisible, so that the information may be displayed more clearly.

By default all features (Circle, Arcs, Points etc.) are placed into display layer 1 and all the dimensions are placed into layer 2.

To change the layer of a feature click on the Measurement Window as shown below:

The layer selected will become the default layer for any further features measured.

To change the layer of a dimension right click on the dimension to bring up the Dimension Details window:

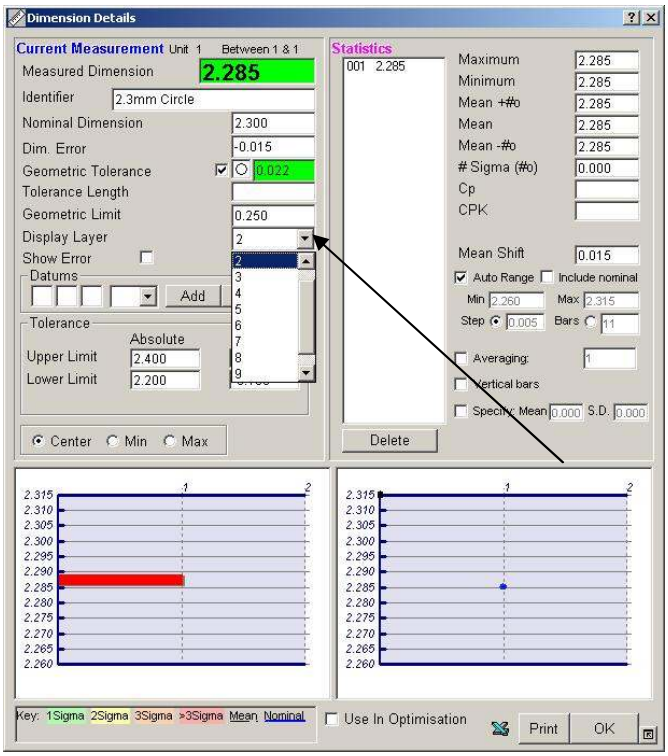


The layer selected will become the default layer for any further features measured.

www.EngineeringBooksPdf.com

Click here to select display layer for a dimension

To change the layer of a dimension right click on the dimension to bring up the Dimension Details window:

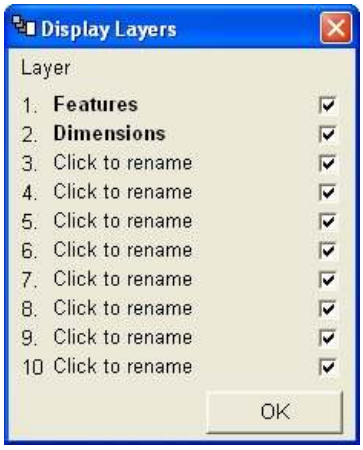


Click on the black arrow adjacent to ‘Display Layer’ and select the desired layer. The layer selected will become the default layer for any further features measured.

To turn layers on and off click on the ‘Display Drawing Layers’ button:

This will bring up the following window:

www.EngineeringBooksPdf.com



Simply clicking on them and typing in the new name can change all of the layer names.

The default layers for both the features and dimensions are shown in bold type. You cannot switch these layers off. If you try to do this, the following warning will appear:

To change the default layer right click on the desired layer to bring up the following menu:

Having changed the default layer you will now be able to switch the original layer off.

www.EngineeringBooksPdf.com

47.0 CHANGE HISTORY

Date of Change	Description of Change		Changed by
12-Oct-07	Chapter 37 'Move via updated		CJH
12-Oct-07	Chapter 10.1.3 Compensated point graphic changed		CJH
30-Oct-07	Chapters 11 to 47 updated		CJH
19-Oct-09	5.2 Datumming the Probe probe length up dated.		CJH
21-Apr-10	Address on front cover and header updated		CJH
18-Nov-10	Choosing between measured & geometric display option added		CJH
19-Nov-10	Update geometric run-out and total run-out		CJH

www.EngineeringBooksPdf.com