Display

Keypad

Axis
Label
Displays

Axis
Displays

Function Display

Axis
Select
Keys

Function
Display

Units Key

Mode Key

Numeric
Keypad

Navigation
Keys

Sleep Key

Setup Key

Clear Entry

Enter
(Store Changes)
<table>
<thead>
<tr>
<th>Specifications</th>
<th>1-1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Electrical</td>
<td>1-1</td>
</tr>
<tr>
<td>Physical</td>
<td>1-1</td>
</tr>
<tr>
<td>Environmental</td>
<td>1-1</td>
</tr>
<tr>
<td>Input</td>
<td>1-2</td>
</tr>
<tr>
<td>Resolutions</td>
<td>1-2</td>
</tr>
<tr>
<td>Connections</td>
<td>2-1</td>
</tr>
<tr>
<td>Cable connections</td>
<td>2-1</td>
</tr>
<tr>
<td>Mountings</td>
<td>3-1</td>
</tr>
<tr>
<td>Arm mounting (non-adjustable)</td>
<td>3-1</td>
</tr>
<tr>
<td>Arm mounting (adjustable)</td>
<td>3-1</td>
</tr>
<tr>
<td>Face mounting (adjustable)</td>
<td>3-1</td>
</tr>
<tr>
<td>Lathe mounting (adjustable)</td>
<td>3-2</td>
</tr>
<tr>
<td>Lathe mounting (adjustable)</td>
<td>3-2</td>
</tr>
<tr>
<td>Display and keypad</td>
<td>4-1</td>
</tr>
<tr>
<td>Display</td>
<td>4-1</td>
</tr>
<tr>
<td>Using the keypad</td>
<td>4-1</td>
</tr>
<tr>
<td>Set up</td>
<td>5-1</td>
</tr>
<tr>
<td>Setup mode</td>
<td>5-1</td>
</tr>
<tr>
<td>Machine type</td>
<td>5-2</td>
</tr>
<tr>
<td>Encoder</td>
<td>5-2</td>
</tr>
<tr>
<td>Radius / diameter</td>
<td>5-3</td>
</tr>
<tr>
<td>Direction</td>
<td>5-3</td>
</tr>
<tr>
<td>Error compensation</td>
<td>5-3</td>
</tr>
<tr>
<td>RS232 options</td>
<td>5-9</td>
</tr>
<tr>
<td>Axis summing</td>
<td>5-12</td>
</tr>
<tr>
<td>Zero approach</td>
<td>5-12</td>
</tr>
<tr>
<td>Taper display axis</td>
<td>5-12</td>
</tr>
<tr>
<td>Reset</td>
<td>5-13</td>
</tr>
<tr>
<td>Store</td>
<td>5-13</td>
</tr>
<tr>
<td>Standard functions</td>
<td>6-1</td>
</tr>
<tr>
<td>Setting the datum for each axis</td>
<td>6-1</td>
</tr>
<tr>
<td>Digifind</td>
<td>6-1</td>
</tr>
<tr>
<td>Centerfind</td>
<td>6-2</td>
</tr>
<tr>
<td>Special functions</td>
<td>7-1</td>
</tr>
<tr>
<td>Mill functions</td>
<td>7-1</td>
</tr>
<tr>
<td>Lathe functions</td>
<td>7-1</td>
</tr>
<tr>
<td>Generic functions</td>
<td>7-1</td>
</tr>
<tr>
<td>Menu function</td>
<td>7-2</td>
</tr>
<tr>
<td>Mill functions</td>
<td>8-1</td>
</tr>
<tr>
<td>Bolt hole circle</td>
<td>8-1</td>
</tr>
<tr>
<td>Arc contouring</td>
<td>8-3</td>
</tr>
<tr>
<td>Line hole</td>
<td>8-5</td>
</tr>
<tr>
<td>Polar coordinates</td>
<td>8-7</td>
</tr>
<tr>
<td>Lathe functions</td>
<td>9-1</td>
</tr>
<tr>
<td>Taper</td>
<td>9-1</td>
</tr>
<tr>
<td>Tool offsets</td>
<td>9-2</td>
</tr>
<tr>
<td>Summing</td>
<td>9-4</td>
</tr>
<tr>
<td>Vectoring</td>
<td>9-5</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>--------------------</td>
<td>------</td>
</tr>
<tr>
<td>Generic functions</td>
<td>10-1</td>
</tr>
<tr>
<td>Sub-datums</td>
<td>10-1</td>
</tr>
<tr>
<td>Jobs</td>
<td>10-3</td>
</tr>
<tr>
<td>Troubleshooting</td>
<td>11-1</td>
</tr>
<tr>
<td>Cleaning</td>
<td>12-1</td>
</tr>
</tbody>
</table>
This chapter details the specifications for the C80.

**Electrical**

**EMC compliance**
- BS EN 61000-6-4:2001
- BS EN 61000-6-2:2001

**Power supply unit (supplied)**
- 100 – 240V (47 – 63Hz)
- External switch-mode
- Conforms to Low Voltage Directive
- EN 60 950-1:2001

**Physical**

**Height**
- 265mm (10.43”)

**Width**
- 180mm (7.09”)

**Depth**
- (not including connectors)
- 50mm (1.97”)

**Weight**
- 2.9kg (6.38lb)

**Environmental**

**Operating temperature**
- 0 to 45°C

**Storage temperature**
- −20 to 70°C

**Environmental conditions**
- Indoor Use, IP20 (IEC 529)

**Relative humidity**
- Maximum 80% for temperatures up to 31°C decreasing linearly to 33% at 45°C
Specifications

Disposal
At the end of its life, you should dispose of the C80 system in a safe manner applicable to electronic goods.

*Do not burn.*

The casework is suitable for recycling. Please consult local regulations on disposal of electrical equipment.

Input
Spherosyn or Microsyn encoders.

Resolutions

**Spherosyn or Microsyn 10**
(menu selection)
- 5µm (0.0002")
- 10µm (0.0005")
- 20µm (0.001")
- 50µm (0.002")

**Microsyn 5**
(menu selection)
- 1µm (0.00005")
- 2µm (0.0001")
- 5µm (0.0002")
- 10µm (0.0005")

Spherosyn and Microsyn are registered trademarks of Newall Measurement Systems Limited.

Newall Measurement Systems Limited reserves the right to make changes to this document.
Connections

This chapter shows the cable connections for the C80.

Cable connections

- **Power Indicator Lamp**
- **2.5mm Power Inlet**
  - 15 Vdc ±10% @1A
- **Equipotential Terminal**
  - (Should be grounded to machine, using the lead provided)
- **RS232 Connection**
- **Cable Anchor**
  - (The power cable should be secured)

- **Encoder Inputs**
  - 9-way ‘D’
  - (All plugs should be screwed finger tight)

- **Blanking Plug**
  - (Do not remove these, as they prevent you accidentally connecting incorrect encoders)
You can only use the C80 with Newall Spherosyn and Microsyn analogue encoders.

You need to ensure that:

• You secure all the cables to prevent the connectors from dropping into hazardous positions (for example the floor or coolant tray) when you unplug them.

• You route all the cables to prevent them from being caught on moving parts.

• The C80 is grounded to the machine, using the braided grounding lead provided, before you turn on the machine supply.

• You turn off the power by disconnecting the power supply connector, before you connect the encoder.

Do not connect this unit directly to the mains supply.

If your Newall encoder is not fitted with a D-type connector, then you can buy an adaptor cable (part no. 307-80980). Contact your supplier for details.
This chapter shows the various options for mounting the C80. You can mount the C80 in a variety of ways. The one that you use depends on the mounting assemblies that you purchased with the unit:

**Arm mounting (non-adjustable)**

Arm Mounting Assembly
UK part no. 294-35675
US part no. FX0000SS-N

M10 stud & fixings, supplied as standard

**Arm mounting (adjustable)**

Arm Mount Adjustable Pivot Assembly
Part no. 600-80290

Mounting Arm Assembly
UK part no. 294-35675
US part no. FX0000SS-N

**Face mounting (adjustable)**

Face Mount Adjustable Pivot Assembly
Part no. 600-80270
Mountings

Lathe mounting (adjustable) with arm assembly

Arm Mount Adjustable Pivot Assembly
Part no. 600-80290

Lathe Carriage Mount with Arm Assembly
UK part no. 600-80310
US part no. FX000LCM-N

Lathe Headstock with Arm Assembly
UK part no. 600-80300
US part no. FX000LHM-N

Lathe mounting (adjustable)

Lathe Mount Adjustable Pivot Assembly
Part no. 600-80280

Lathe Carriage Assembly
Part no. 294-61695

Lathe Headstock Mount Assembly
Part no. 294-60675
This chapter explains how to interpret the display and use the keypad.

You should refer to the diagrams at the front of this manual for information on the layout of the display and keypad.

Display

Axis displays
The three Axis displays normally show the positions of the X, Y and Z axes.

The three Axis Label displays normally show X, Y and Z, (in Lathe Mode X, Z and Z').

Function display
While any axis is moving, the Function display shows the Feedrate of the fastest moving axis. The feed indicator next to the display is illuminated. The feedrate is displayed in mm/sec (to a resolution of 0.5) or inches/min (to a resolution of 1.0).

If the lathe indicator is illuminated, then the lathe functions are available; if the mill indicator is illuminated, then the mill functions are available; if the lathe and mill indicators are illuminated, then both the mill and lathe functions are available.

During Setup and Special Functions, the displays may show information other than that described on this page. See the Set up and Special functions chapters of this manual for further information.

Using the keypad
You use the keypad to enter data and control information for the C80. The keys can have different functions depending on the operating mode, but in normal operation the keys work as follows:

Unit Key. Toggles the display units between inches and millimetres.

Mode Key. Toggles the C80 between Absolute Mode and Incremental Mode.

Absolute mode
In this mode, the C80 displays the positions of the three axes relative to a fixed datum.

Incremental mode
In this mode, the C80 displays each position relative to the last position. This is also known as point-to-point use.

At the beginning of each working session, set the datum in Absolute Mode, then switch the C80 to Incremental Mode. By using the C80 in this way, you can return the machine to its absolute datum at any time, simply by switching back to Absolute Mode.
Sleep mode
Press the [Sleep Key] to temporarily turn off the displays and the keypad.

While the unit is in Sleep Mode, all the settings are preserved, but the positions of the three axes are updated.

If you move any of the axes while in Sleep Mode, the centre display shows ‘d i S P L A C d’.

If someone touches any of the keys, the centre display shows ‘t o u c h E d’.

See the Set up and Special functions chapters of this manual for more explanation of the use of the Navigation and Function Keys.
This chapter describes how to set up the C80.

Setup mode
Normally, you only need to set up the C80 once. You may find that the factory default settings are suitable so you do not need to change them.

To enter Setup Mode:
1. Leave any Special Function that is running.
2. Press [set up]. The centre display shows ‘SEt Up’.
3. Press the up or down navigation key to cycle through the list of options.

The following table lists the options. The following sections describe each of the options in detail.

<table>
<thead>
<tr>
<th>Option</th>
<th>Default</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Machine Type</td>
<td>Generic</td>
<td>gEnEr ic</td>
</tr>
<tr>
<td>Encoder Type</td>
<td>all axes: Spherosyn</td>
<td>SPHER oSn</td>
</tr>
<tr>
<td>Encoder Resolution</td>
<td>all axes: 0.005mm</td>
<td>0.005</td>
</tr>
<tr>
<td>Radius / Diameter</td>
<td>all axes: Radius</td>
<td>rAD</td>
</tr>
<tr>
<td>Direction</td>
<td>all axes: 1</td>
<td>dir. 1</td>
</tr>
<tr>
<td>Error Compensation</td>
<td>all axes: Off</td>
<td>Err OFF</td>
</tr>
</tbody>
</table>

The options available depend on the setting of other options. For example, the Zero Approach Limit option is not present if Zero Approach is off.
The following instructions are important and should be followed carefully.

When you have finished setting all the options:
1. Press the down navigation key until ‘St o r E ’ appears in the middle display.
2. Press [ent] to store any changes that you have made.
   The middle display shows ‘St o r E d ’ for a few seconds as your settings are stored.

The C80 leaves Setup Mode.

Alternatively, you can press [set up] at any time to leave the Setup Mode and abandon any changes.

**Machine type**

You use this option to choose whether the special functions for Mill or Lathe are available.

There are three possible settings:

<table>
<thead>
<tr>
<th>Generic</th>
<th>g E n E r i c</th>
<th>all functions available</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mill</td>
<td>m i L L</td>
<td>mill functions only</td>
</tr>
<tr>
<td>Lathe</td>
<td>L A t H E</td>
<td>lathe functions only</td>
</tr>
</tbody>
</table>

Press the Select Key next to the ‘ t ’ to cycle through these settings.

**Encoder**

The Encoder settings must match the actual encoder in use, or the C80 will not display correctly.

**Encoder type**

There are three possible settings for each axis:

<table>
<thead>
<tr>
<th>Encoder type</th>
<th>Spherosyn</th>
<th>Microsyn 10</th>
<th>Microsyn 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spherosyn</td>
<td>SPHEr o S n</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Microsyn 10</td>
<td>u S n  10</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Microsyn 5</td>
<td>u S n  5</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Press the Select Key next to the ‘X’, ‘Y’ or ‘Z ’ to cycle through these settings.

**Encoder resolution**

The Resolution settings available for each axis depend on the encoder type and the [in/mm] setting.

<table>
<thead>
<tr>
<th>µm</th>
<th>mm</th>
<th>in</th>
<th>Spherosyn 10</th>
<th>Microsyn 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.001</td>
<td>0.00005</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>2</td>
<td>0.002</td>
<td>0.0001</td>
<td></td>
<td>X</td>
</tr>
</tbody>
</table>
### Radius / diameter

The **Diameter** setting is useful for lathes, and other turning applications to display the diameter reading rather than the radius.

When you select the **Diameter** setting the **C80** displays double the actual movement on any axis.

There are two settings for each axis:

<table>
<thead>
<tr>
<th>Radius</th>
<th>r</th>
<th>Ad</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter</td>
<td>d</td>
<td>i</td>
</tr>
</tbody>
</table>

Press the **Select Key** next to the ‘X’, ‘Y’ or ‘z’ to cycle through these settings.

### Direction

You use the **Direction** setting to match the **C80** to the actual direction of travel of any axis.

There are two settings for each axis: ‘dir. 0’ and ‘dir. 1’.

Press the **Select Key** next to the ‘X’, ‘Y’ or ‘z’ to cycle through these settings.

*The Direction setting is arbitrary. You should set it to whichever option makes most sense for the machine.*

*The Direction depends on where the scale is mounted.*

### Error compensation

Your digital readout (DRO) system helps you to improve productivity. It decreases the number of scrapped parts, as you no longer have to be concerned about making mistakes related to counting the revolutions on the dials. Your DRO system also helps to eliminate some errors related to ballscrew backlash.

Your DRO system will operate to its published accuracy, provided all components are in working order and properly installed. Field calibration is not necessary.

Accuracy problems with machined parts may be caused by machine error, DRO system error, or a combination of both. The first step in determining the source of error is to check the DRO system. You do this by comparing the movement of the Newall reader head to the position reading shown on the display. You

---

<table>
<thead>
<tr>
<th>µm</th>
<th>mm</th>
<th>in</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>0.005</td>
<td>0.002</td>
</tr>
<tr>
<td>10</td>
<td>0.01</td>
<td>0.0005</td>
</tr>
<tr>
<td>20</td>
<td>0.02</td>
<td>0.001</td>
</tr>
<tr>
<td>50</td>
<td>0.05</td>
<td>0.002</td>
</tr>
</tbody>
</table>

Press the **Select Key** next to the ‘X’, ‘Y’ or ‘z’ to cycle through the available settings for each axis.
need a high accuracy standard, such as a laser interferometer. You can use a dial indicator to check short distances, but a laser provides the best results. If you have to use a dial indicator, be sure it is the highest available accuracy.

To check the accuracy of the DRO system:

1. Place the target of the laser or the needle of the dial indicator directly on the Newall reader head. It is absolutely critical that you take the readings directly from the Newall reader head. If you have to use a dial indicator, be sure that the needle of the indicator is perpendicular to the reader head and not angled. If you take readings anywhere else on the machine, machine errors may distort the results.

2. When the reader head moves, the movement registers on the laser / indicator and DRO display.

3. Set the laser / dial indicator and DRO display position displays to 0.

4. Make a series of movements and compare the position readings between the laser / dial indicator and the DRO display. If the readings match within the accuracy specified, then you know that the DRO system is operating properly. If this is the case, you can proceed to the next step: evaluating the machine errors. If the readings do not match, you must repair the DRO system before proceeding with error compensation.

To evaluate machine errors:

1. Put the laser target / dial indicator on the part of the machine where the machining is done.

2. Make a series of movements and compare the position readings between the laser / dial indicator and the DRO display. The difference between the laser / dial indicator reading and the reading on the DRO display is your machine error.

3. Plot the machine error along the entire axis of travel to determine the nature of the error. If it is a linear error, you can use linear error compensation. If the error is not linear, you should use segmented error compensation.

There are many types of machine error, including pitch, roll, yaw, flatness, straightness, and Abbe error. The diagrams below demonstrate these errors.
Error compensation

Way errors

There are three settings for each axis:

<table>
<thead>
<tr>
<th>Setting</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Off</td>
<td>Err OFF</td>
</tr>
<tr>
<td>Segmented Compensation</td>
<td>SEG Err</td>
</tr>
<tr>
<td>Linear Compensation</td>
<td>Lin Err</td>
</tr>
</tbody>
</table>

Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to cycle through these settings.

Once you have set the error compensation you want to use on all the axes, press the down navigation key. The middle display changes to ‘Err Set’.

If you set one or more axes to Segmented Error Compensation, or Linear Error Compensation, then your next set up option is to configure the compensation for each of those axes.

If you apply Error Compensation, then you must ensure that it is absolutely correct. If it is not correct, then errors could be increased rather than reduced.

Once you have set up the Error Compensation, we advise you to check its effect in normal operation.

Segmented error compensation

In this mode, you can break down the scale travel for each axis into as many as 99 user-defined segments, with each segment having its own correction factor. The correction factors are
calculated by the C80 by comparison against known standards that you supply.

When you apply power, the display for any axis that is set to use **Segmented Compensation** shows ‘r E S E t’.

If the machine has not been moved since the power was turned off, press [ce], and the C80 restores the last positions that were recorded.

Alternatively, you can set each axis close to the **Reference Point**, to within: 6.3mm (0.25") for a Spherosyn encoder or 2.5mm (0.1") for a Microsyn encoder, and press the **Select Key** next to the ‘X’, ‘Y’ or ‘z’. The C80 re-establishes alignment with the correction parameters.

You need not apply Segmented Compensation over the entire scale length. You can apply it to a length of high importance, or to just one segment.

To ensure that you apply the correct compensation, you should always reset the datum when you turn on the C80.

If you set one or more axes to Segmented Error Compensation, then you need to carry out the following procedure to configure the compensation for each of those axes.

**Identifying the correction parameters**

The scale travel is broken down into a number of segments that you define, each with its own correction factor, measured against a high-accuracy standard. You need to identify the following parameters:

You measure each **Correction Point** with respect to the **Starting Point**, zero. You usually set this close to one end of the scale. You can set the **Reference Point** anywhere along the scale. It does not need to coincide with either the absolute datum or any of the correction points. However, you may find it convenient to make the absolute datum and the reference point the same.

**Setting the correction points**

As you follow the procedure you must ensure that you always approach the **Starting Point**, **Correction Points** and **Reference Point** from the same direction. If you do not, then the size of the tool or probe renders the measurement inaccurate.

1. Set one or more axes to **Segmented Compensation** as described in Error compensation on page 5-3. The display shows ‘E r S E t’.
2. Press the Select Key next to the ‘X’, ‘Y’ or ‘Z’ to enter the setup procedure for that axis.
   The display changes to ‘SEt ZEro’.

3. Set the machine to the point you have chosen as your Starting Point, and zero the high-accuracy standard at this point.

4. Press [ent].
   The display changes to ‘goto 1’.

5. Set the machine to the point you have chosen to be Correction Point 1.

6. Press [ent].
   The display changes to ‘Ent Sd 1’.

7. Enter the distance from the Starting Point, as measured by the standard.
   The C80 calculates and displays the correction factor for this point.

8. Press the down navigation key to go to the next point.

9. Repeat steps 5 to 8 for each Correction Point.

10. When you have entered all the correction points, press [abs/inc].
    The display changes to ‘goto O r Ef’.

11. Set the machine to the point you have chosen as the Reference Point.

12. Press [ent].
    The display returns to ‘Err SEt’.

13. If required, press the Select Key next to ‘X’, ‘Y’ or ‘Z’ to enter the error compensation for another axis.

You can define up to 99 segments per axis.

To use Segmented Error Compensation, you need to have a high accuracy standard, such as a laser measuring system.

Segmented Error Compensation initially defaults to Off, with no points set.
If you set Segmented Error Compensation to Off after you have set the Correction Points, then the data is retained, but not applied. When you next set the Segmented Error Compensation to On, the data is re-applied.

You must carry out this procedure in strict sequence, and in full, for it to be valid.

You can press the Select Key at steps 1 to 8 to display the current uncorrected position relative to the Starting Point.

Do not worry about the direction of the standard measurement. For example, 678.9 and −678.9 are treated the same.

You can press [ce] to clear an entry one character at a time.
Once you have pressed [ent] to complete an entry, you can press [ce] to take you back one step at a time.

**Linear error compensation**

In this mode, you can apply a single constant correction factor for each axis to all displayed measurements. You calculate the correction factor, and specify it in parts per million (ppm). The values can be between $-9999$ and $+9999$.

In this mode a single constant correction factor for each axis is applied to all displayed measurements.

If you set one or more axes to **Linear Error Compensation**, then you need to carry out the following procedure to configure the compensation for each of those axes.

**Calculating the correction factor**

As you follow the procedure you must ensure that you either use a stepped standard, and approach each edge from the same direction; or if you must approach each edge from opposite directions, then subtract the width of the tool or measuring probe from the value displayed on the **C80**.

For example: To check the scale against a standard which is exactly 500mm wide:

1. Set the tool or probe to one edge of the standard, and press the **Select Key** for the axis that you need to correct. The display shows ‘0.000’.

2. Set the tool or probe to the other edge of the standard. The display shows ‘499.800’.

3. Calculate the correction factor:
   
   \[
   \text{error} = 500.000 - 499.8 = 0.2\text{mm} \\
   \text{Correction Factor} = \frac{\text{error}}{\text{standard}} = \frac{0.2}{500} \times 1,000,000 = +400 \text{ ppm} \text{ (parts per million)}
   \]

For this example you need to increase the value displayed on the **C80** to match the standard, as this is a positive correction factor. If the display had shown 500.2 for the same standard, the correction factor would be negative, $-400$ ppm.
Setting the correction factor

To set the correction factor:

1. Set one or more axes to Linear Error Compensation as described in Error compensation on page 5-3. The display shows ‘Err Set’.
2. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to enter the setup procedure for that axis. The display shows ‘LC 0’, or a previously entered value. For example: Press [4] [0] [0] [±] [ent] to enter a Correction Factor of −400 ppm.
4. If required, press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to enter the setup procedure for another axis.

You cannot establish the Correction Factor while in Setup Mode. You need to carry out the measurements in Normal Operating Mode, then enter Setup Mode to set the Correction Factor.

The value you set must be in the range −9999 to 9999.

If you make a mistake while entering a number, pressing [ce] clears the entry one character at a time.

RS232 options

RS232 was added as a standard feature to the C80 in March 2005.

The C80 DRO can offer basic RS232 communications via a dedicated hardware RS232 compatible port.

You select the baud rates for communications from the following options:

300, 1200, 2400, 4800, 9600, 14400, 19200, 38400

There are three RS232 modes:

No RS232. This is the default condition. All the RS232 functions are disabled and there are no outputs.

Continuous Output. From the menu structure you define the frequency of the output. The options are 0.1 – 60.0 in steps of 0.1 second.

Keyed Output. For this option the axes data is transmitted when you press the [ent] key, without having pressed the preceding keys. Thus you do not need to use a function key.

Output data format

The output data from the RS232 is as follows:

- The current axis data for the axes available on the system are transmitted
Set up

- For two axis systems, only two axes of data will be transmitted.
- The data packet structure of 12 characters is defined as follows:

```
A : 0 0 0 0 0 0 0 0 CR LF
```

The Axis ID is the character shown in the axis 15-segment display at the time of printing. The exception is that for three-axis Lathe applications a lower case ‘z’ is used to denote the compound Z’-axis. An upper case ‘Z’ is used to denote the standard Z-axis.

**System settings**

- Baud rate = Configurable CR (300, 1200, 2400, 4800, 9600, 14400, 19200, 38400)
- Data bits = 8
- Parity bit = Configurable (Even, Off, None)

Stop bits = 1
Flow control = None

**Default system settings**

- Serial option = None
- Serial rate = 1.0 (i.e. once per second)
- Baud rate = 9600
- Parity bit = None

**Connections**

You connect the RS232 to the C80 via a 15-pin D-type connector at the rear of the display.
Axis summing

All other connections are reserved and should be left unconnected or the unit may be damaged.

Operation
You configure the RS232 output from the setup menu, using the following settings:

- ‘SERIAL’ refers to the mode of operation. This can be ‘CONSTANT’, ‘SINGLE’ or ‘NONE’. You press the left or right navigation key to cycle through the options.

- ‘S E r r a t e ’ refers to the rate at which RS232 data is generated when in constant mode. It shows a value (in seconds) in the second axis window. You press the arrow button next to the value to change the serial rate. You can select values in the range 0.1 – 60.0 in 0.1s increments. The maximum serial rate is limited by the baud rate, as shown in the following table.

<table>
<thead>
<tr>
<th>Baud Rate</th>
<th>Maximum Serial Rate (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>300</td>
<td>2.0</td>
</tr>
<tr>
<td>1200</td>
<td>1.0</td>
</tr>
<tr>
<td>2400</td>
<td>1.0</td>
</tr>
<tr>
<td>4800</td>
<td>1.0</td>
</tr>
<tr>
<td>9600</td>
<td>0.5</td>
</tr>
</tbody>
</table>

- ‘b d r A t E’ relates to the baud rate of the RS232 communications. The second axis window shows the current value. You press the left or right navigation key to select a value.

- ‘P a r i t y’ refers to the parity mode of the RS232 communications. The second axis window shows the current setting. This can be ‘E U E n’, ‘O d d’ or ‘n O n E’. You press the left or right navigation key to scroll through the options.

Axis summing
This setting works in conjunction with the Summing function.

Two of the axis displays show ‘Addition’.

Press the Select Keys to switch between the two settings, X,Z and Z,Z’.
**Zero approach**
This flashes the Axis Label display when one or more axes are approaching zero.

For example: If you set the Zero Approach for the X axis, with a Zero Approach Limit of 1.25, then the axis label display flashes for values from ‘- 1.250X’ to ‘1.250X’. When the axis is within 0.05mm (0.002”) for a Spherosyn encoder or 0.025mm (0.001”) for a Microsyn encoder, the display stops flashing.

**Zero approach on / off**
There are two settings for each axis:

<table>
<thead>
<tr>
<th>Zero Approach On</th>
<th>Z E r 0 0 n</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zero Approach Off</td>
<td>Z E r 0 0 F</td>
</tr>
</tbody>
</table>

Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to cycle through these settings.

**Zero approach limit**
You use this setting to choose how close to zero the axis needs to be for the display to flash.

To set the Zero Approach Limit:
1. Set the Zero Approach On.

2. Press the down navigation key. The displays for the selected axes change to ‘0 0 0 0’ or a previously entered value.

3. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to choose which axis to edit.

   For example: To enter a limit of 1.25, press [1] [.] [2] [5] [ent].

4. If necessary, press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to enter the limit for another axis.

**Taper display axis**
This setting works in conjunction with the Taper function.

One of the axis displays shows ‘t A P E r  O n ’ while the other two displays show ‘t A P r  O F F ’.

Press the Select Keys to choose which axis displays the Taper function.

**Reset**
This restores all settings to their factory defaults, and you should, therefore, only use it if absolutely necessary.

The middle display shows ‘r E S E t ’.

Press [ent] or the Select Key next to the ‘r ’ to select the Reset function.
While all the stored settings are being erased, the top display shows ‘ClEArIng’, and the middle display shows ‘0’, ‘0 0’, etc.

When Reset has finished the middle display returns to ‘rESEt’.

The C80 remains in Setup Mode.

*Use Reset with caution, as you will lose all your stored settings.*

Reset takes approximately 15 seconds.

---

**Store**

⚠️ *This is an important section, you should read it carefully.*

This stores all your settings, then returns to Normal Operating Mode.

The middle display shows ‘St or E’.

Press [ent] or the Select Key next to the ‘S’ to select the Store function.

The middle display shows ‘St or Ed’ for a few seconds, as your settings are stored.

The C80 leaves Setup Mode.

Alternatively, you can press [set up] at any time to leave Setup Mode and abandon any changes.
| Set up |
This chapter describes the standard functions for the C80.

**Setting the datum for each axis**

**Setting the zero**
To zero one display at the current position, press the Select Key by the axis to set it to zero.

All readings are now relative to this new zero point.

*Using Zero redefines the datum. You cannot restore the old datum.*

**Setting the preset**
To preset one display to a known value:

1. Press [preset].
2. Press the Select Key by the axis to preset it.
3. Enter the value.

*If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.*

**Recalling the last value**
To quickly recall the last preset value for an axis:

1. Press [recall].
2. Press the Select Key to preset the axis.

All readings are now relative to this new value.

**Digifind**
If you lose a datum, either due to movement following a power failure, or after you have entered a fixed point by mistake, you can re-establish it using Digifind.

**Using Digifind**
The absolute datum for each axis should be marked permanently on the machine.

1. Set the axis close to the marked datum, to within:
   - 6.3mm (0.25") for a Spherosyn encoder or
   - 2.5mm (0.1") for a Microsyn encoder.
2. Switch the C80 to Absolute Mode.
   Digifind only works in absolute mode.
3. Press [ref].
4. Press the Select Key to restore the axis.

The display updates to show the exact distance from the datum.
Centerfind
Centerfind halves the distance displayed on the selected axis, so that you can find the centre of a workpiece. It works in either Absolute or Incremental Mode.

Using Centerfind
For example, to find the centre of a workpiece that is 100mm wide:

1. Set the tool to one edge of the workpiece.
2. Press the Select Key to centre the axis.
   The display shows ‘0.000’.
3. Set the tool to the other edge of the workpiece.
   The display shows ‘100.000’.
4. Press [1/2].
   The display shows ‘0’ in all axes.
5. Press the Select Key to centre the axis.
   The display now shows ‘50.000’.
6. Move the tool until the display shows ‘0.000’.

This is the centre of the workpiece.

No compensation for tool diameter (width) is provided for this function.

Using the centerfind feature in incremental mode preserves the absolute mode datum setting.
This chapter describes the C80’s special functions.

In addition to the Standard Functions described on page 6-1, the C80 has a number of inbuilt Special Functions, that you access using the [F1], [F2] and [F2+] keys.

Most Special Functions work specifically in the Mill or Lathe modes, but the Generic functions can work with either.

Most Special Functions require only one function key to operate. You can allocate them to either [F1] or [F2].

The functions marked [F2] require two function keys. You can only allocate these to [F2] and [F2+].

The Generic option also includes all the Mill and Lathe Special Functions.

Each of the Special Functions listed here is described in detail later in the Mill functions, Lathe functions and Generic functions chapters.

### Mill functions
The Special Functions in Mill mode are:

<table>
<thead>
<tr>
<th>Special Function</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bolt Hole Circle</td>
<td>b O L t</td>
</tr>
<tr>
<td>Arc</td>
<td>A r C</td>
</tr>
</tbody>
</table>

### Lathe functions
The Special Functions in Lathe mode are

<table>
<thead>
<tr>
<th>Special Function</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Line Hole</td>
<td>L i n E</td>
</tr>
<tr>
<td>Polar Coordinates</td>
<td>P o L A r</td>
</tr>
</tbody>
</table>

### Generic functions
The Special Functions in Generic mode are

<table>
<thead>
<tr>
<th>Special Function</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sub Datum and Job Numbers</td>
<td>S d m</td>
</tr>
</tbody>
</table>

Special Function Display

Lathe functions

<table>
<thead>
<tr>
<th>Special Function</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool Offsets</td>
<td>t o o l</td>
</tr>
<tr>
<td>Taper</td>
<td>t a p e r</td>
</tr>
<tr>
<td>Summing</td>
<td>s u m</td>
</tr>
<tr>
<td>Vector</td>
<td>V e c t o r</td>
</tr>
</tbody>
</table>

Generic functions

<table>
<thead>
<tr>
<th>Special Function</th>
<th>Display</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sub Datum and Job Numbers</td>
<td>S d m</td>
</tr>
</tbody>
</table>
**Menu function**

You can only use two *Special Functions* at a time.

To find out which function is allocated to each key:

1. Press [menu] to see the Menu.
   The display shows:
   `bolt HOL 1`
   `Arc 2`
2. Press [menu] again to turn the Menu off.

**Using a function**

To use a function press [F1], [F2] or [F2+], according to the instructions given later in this guide.

**Allocating a function to a key**

To allocate a function to a key:

1. Press [menu].
2. Press the Select Key next to the ‘1 ’ or ‘2 ’ to choose which function key to edit.
3. Press the up or down navigation key to cycle through the list of *Special Functions*.
4. Press [ent] to allocate the selected *Special Function* to the function key.

*If certain functions are running when you press [menu], then in place of the function name, the display shows 'turn Off'. If you want to allocate a function, press [menu] again to turn the Menu off. If you want to allocate a different function to that function key, then you need to turn the function off before trying again.*
This chapter describes the special functions available in Mill mode.

Mill functions are available when you have configured the C80 for either **Mill** or **Generic** operation.

**Bolt hole circle**

*You cannot use arc contouring and bolt hole circle functions at the same time.*

*This function is also known as Pitch Circle Diameter (PCD).*

This function calculates the locations of the holes, given the following parameters:

1. Plane \((X,Y; X,Z\text{ or } Y,Z)\)
2. Circle Centre location
3. Circle Diameter
4. Number of Holes (up to 99)
5. Starting Angle (measured anti-clockwise from three o’clock)
Mill functions

Setting the parameters

1. Press [F1] or [F2] to turn the function on. For three axis units only. The function display shows ‘p’, and the axis displays show the Plane in which the holes are to be machined.
2. Press the up or down navigation key to cycle through the three settings X,Y; X,Z or Y,Z.
3. Press the right navigation key to move to the next step. The function display shows ‘c’, and the axis displays show the coordinates of the Circle Centre.
4. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’, to edit each value as required.
5. Press the right navigation key to move to the next step. The function display shows ‘d’, and the top axis display shows the Circle Diameter.
6. Enter a new value if required. For example: Press [1] [5] [0] [ent] to enter a diameter of 150.
7. Press the right navigation key to move to the next step. The function display shows ‘n’, and the top display shows the Number of Holes.
8. Enter a new value if required. For example: Press [5] [ent] if you want to machine 5 holes.
9. Press the right navigation key to move to the next step. The function display shows ‘a’, and the top display shows the Starting Angle.
10. Enter a new value if required. For example: Press [1] [8] [ent] to enter a value of 18 degrees.
11. Press the right navigation key to finish setting the parameters. The function display shows ‘0 1’.

Machining the holes

The two axis displays for the selected plane now show the distance to the first hole.

1. To position the tool ready for machining the hole, move the axes until both displays show zero. The function display shows the number of the hole to be machined.
2. Press the left or right navigation key to move between the holes, or enter the hole number. For example: Press [4] [ent] to move directly to hole 4.
3. When all the holes have been machined, press [F1] or [F2] to turn the function off.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.
Once you have pressed [ent] to complete an entry, you can press the navigation keys to move backwards and forwards one step at a time.

To turn the function off, finish making any entry, then press the function key again.

The axis that is not involved in the Bolt Hole Circle function reads as normal.

**Arc contouring**

You cannot use arc contouring and bolt hole circle functions at the same time.

This function calculates the locations of the points along the line of the arc, given the following parameters:

The maximum number of points is 100 and the angle must be less than 180 degrees.

1. Plane (X,Y; X,Z or Y,Z)
2. Arc Centre location
3. Arc Radius
4. Starting Point
5. Ending Point
6. Tool Diameter

7. Internal or External Cut (machined to the inside or the outside of the arc)

8. Maximum Cut (the smaller the cut, the more points calculated)

**Example**
Setting the parameters

1. Press [F1] or [F2] to turn the function on.
   (For three axis units only)
   The function display shows ‘p’, and the axis displays show the Plane in which the holes are to be machined.

2. Press the up or down navigation key to cycle through the three settings X,Y; X,Z and Y,Z.

3. Press the right navigation key to move to the next step.
   The function display shows ‘C’, and the axis displays show the coordinates of the Arc Centre.

4. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’, to edit each value as required.

5. Press the right navigation key to move to the next step.
   The function display shows ‘r’, and the top display shows the Arc Radius.

6. Enter a new value if required.

7. Press the right navigation key to move to the next step.
   The function display shows ‘s t’, and the axis displays show the coordinates of the Starting Point.

8. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’, to edit each value as required.

9. Press the right navigation key to move to the next step.
   The function display shows ‘e n d’, and the axis displays show the coordinates of the Ending Point.

10. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to edit each value as required.

11. Press the right navigation key to move to the next step.
    The function display shows ‘t d’, and the top display shows the Tool Diameter.

12. Enter a new value if required.
    For example: Press [.5] [ent] to enter a diameter of 0.5.

13. Press the right navigation key to move to the next step.
    The function display shows ‘l e’, and the top axis display shows whether the cut is to be machined to the internal or the external radius of the arc. The display shows Internal ‘r a d - t 00|’ or External ‘r a d t 00|’.

14. Press the up or down navigation key to cycle through the settings.

15. Press the right navigation key to move to the next step.
    The function display shows ‘C u t’, and the top display shows the Maximum Cut.

16. Enter a new value if required.
    For example: Press [1] [5] [ent] to enter a maximum cut of 0.15.
17. Press the right navigation key to finish setting the parameters.
   The function display shows '0 1'.

**Machining the arc**
The two axis displays for the selected plane now show the distance to the Arc starting point.

1. To position the tool ready for machining the arc, (starting point) move the axes until both displays read zero.
   The function display shows the number of the hole to be machined.

2. Press the left or right navigation key to move between the positions.

3. When the Arc machining is complete, press [F1] or [F2] to turn the function off.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.

Once you have pressed [ent] to complete an entry, you can press the left and right navigation keys to move backwards and forwards one step at a time.

If you enter a **Starting Point** or **Ending Point** that is inconsistent with the **Centre** and **Radius** settings, then the **Centre** and **Radius** settings override the inconsistent settings.

The axis that is not involved in the Arc function reads as normal.

You must machine the arc progressively. You cannot jump between points on the arc.

You should move away from the line of the Arc between points to avoid over cutting.

**Line hole**
This function calculates the locations of the holes, given the following parameters:

1. Plane (X,Y; X,Z or Y,Z)
2. Starting Point
3. Line Length
4. Number of Holes (up to 99)
5. Line Angle

The axis that is not involved in the line hole function reads as normal.
Mill functions

Example

1. Press [F1] or [F2] to turn the function on. (For three axis units only) The function display shows ‘p’, and the axis displays show the Plane in which the holes are to be machined.
2. Press the up or down navigation key to cycle through the three settings X,Y; X,Z or Y,Z.
3. Press the right navigation key to move to the next step. The function display shows ‘s t’, and the axis displays show the coordinates of the Starting Point.
4. Press the Select Key next to the ‘X’, ‘Y’ or ‘Z’, to edit each value as required.
5. Press the right navigation key to move to the next step. The function display shows ‘l e n’, and the top display shows the Line Length.
6. Enter a new value if required. For example: Press [3] [5] [0] [ent] to enter a line length of 350.
7. Press the right navigation key to move to the next step. The function display shows ‘n’, and the top display shows the Number of Holes.
8. Enter a new value if required. For example: Press [9] [ent] if you want to machine 9 holes.
9. Press the right navigation key to move to the next step. The function display shows ‘a’, and the top display shows the Line Angle.
10. Enter a new value if required. For example: Press [2] [0] [ent] to enter a value of 20 degrees.
11. Press the right navigation key to finish setting the parameters.
   The function display shows '01'.

**Machining the holes**

The two axis displays for the selected plane now show the distance to the first hole.

1. To position the tool ready for machining the hole, move the axes until both displays read zero.
   The function display shows the number of the hole to be machined.

2. Press the left or right navigation key to move between the holes, or enter the hole number. For example: Press [4] to move directly to hole 4.

3. When all the holes have been machined, press [F1] or [F2] to turn the function off.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.

Once you have pressed [ent] to complete an entry, you can press the left and right navigation keys to move backwards and forwards one step at a time.

To turn the function off, finish making any entry, then press the function key again.

---

**Polar coordinates**

This function converts the position of two selected axes into Polar coordinates.

The C80 normally uses the Cartesian Coordinate System, in which the position of a point in any plane is defined by two coordinates \((X,Y; X,Z)\) or \((Y,Z)\).

In the Polar Coordinate System, an imaginary line is drawn between the position of the point and the datum. The Polar coordinates displayed are the length of the line \((P)\), and its angle, measured anti-clockwise from three o’clock.
Using the Polar coordinates function

1. Press [F1] or [F2] to turn the function on.
   The axis label displays for two of the axes show ‘P ’ and ‘a ’.

2. Press the left or right navigation key to cycle through the three Plane settings ‘X ’,‘Y ’, ‘X ’,‘Z ’ or ‘Y ’,‘Z ’.

3. Press [ent] to accept the setting.
   The axis that is not involved in the Polar Coordinates function displays as normal.
Lathe functions

This chapter describes the special functions available in Lathe mode.

Lathe functions are available when you have configured the C80 for either Lathe or Generic operation.

The conventional way to set up a lathe is:

X Axis – cross travel
Z Axis – longitudinal travel
Z’ Axis – compound travel.

If you set the Machine Type to Generic, then the axes are labelled:
Axis 1 – X
Axis 2 – Y
Axis 3 – Z

Taper
You use this function to turn or measure a turned, tapered part.

One of the axes displays the Taper angle. This is the angle between the present machine position and the datum, in the X,Z plane.

We recommend that you use this function in Incremental Mode, as it involves changing the datum.

The axes that do not show the Taper angle display as normal.

Setting the parameters
You select which axis is to display the Taper angle in Setup Mode.

To enter Setup Mode:

1. Leave any Special Function that is running.
2. Press [set up].
   The centre display shows ‘set Up’.
3. Press the up or down navigation key to choose ‘t a p e r’.
4. Press the Select Key next to the ‘X’, ‘Y’ or ‘z’ to choose which axis shows ‘t a p e r On’.
5. Press the up or down navigation key to choose ‘s t o r e’.
6. Press [ent] to store the change.
Lathe functions

Using the taper function
1. Touch the tool to one end of the taper.
2. Press the Select Keys next to the ‘X’ and ‘z’, to set the datum.
3. Press [F1] or [F2] to turn the function on.
4. Touch the tool to one end of the taper. The axis display marked ‘a’ shows the taper angle.
5. Press [F1] or [F2] to turn the function off.

Tool offsets
With this function you can program the C80 with Tool Offsets for as many as 99 different tools, to save having to reset the datum every time you change tools.

You should set the C80 to Incremental Mode before using Tool Offsets. By doing this, you can return the machine to its absolute datum, by switching back to Absolute Mode.

This function is designed primarily for use in Lathe Mode, but you can also use it in Generic Mode.

You can only set offsets for the first two axes, which on a lathe are the X and Z axes. In the examples opposite, the X axis is set to the diameter of the part, and the Z axis is zeroed at the face.

The Tool 1 Offset is special, because it is tied to the Machine Datum, as explained below.

Tool set mode
You press [F2] to access this mode, and use it to set the offsets for each tool.

To turn off Tool Set Mode, press [F2].

Setting the Tool 1 Offset in this mode affects the Machine Datum. Similarly, a change to the Machine Datum changes the Tool 1 Offset. We therefore recommend that you set the C80 to Incremental Mode before using this function.

Setting each tool offset in this mode is independent of all the others, so a change to the Machine Datum or to the Tool 1 Offset does not affect the other tool offsets.
Tool offsets

**Tool usage mode**
You press \([F2^+]\) to access this mode, and use it once you have set all the offsets.

To turn off **Tool Usage Mode**, press \([F2^+]\).

*Changing to the Machine Datum while in this mode changes all the offsets. This can be useful if the same set of tools is to be used on parts of varying sizes.*

**Setting the tool offsets**
1. Press \([F2]\) to turn on the **Tool Set Mode**.
2. Press the right navigation key to select the Datum Tool (Generally Tool No. 1). The function display shows the tool number ‘01’.
3. Take a skim cut along the outside diameter of the part or touch the tool to the surface of the part (if cylindrical).
4. Move the tool away from the part, taking care not to move the X axis.
5. Measure the diameter of the part using a suitable gauge.
6. Press the **Select Key** next to the ‘X’ and enter the diameter of the part as measured using the numbers on the keypad, then press \([\text{ent}]\).
7. Take a facing cut or touch the end of the part with the tool.
8. Move the tool away from the part, taking care not to move the Z axis.
9. Press the **Select Key** next to the ‘z’ and press \([\text{ent}]\) to zero the axis. You have now established the **Tool Offsets Datum**.
10. Press the right navigation key to move to the next tool.
11. Touch the tool to the surface of the part.
12. Move the tool away from the part, taking care not to move the X axis.
13. Measure the diameter of the part using a suitable gauge.
14. Press the **Select Key** next to the ‘z’ and enter the diameter of the part as measured, using the numbers on the key pad, and then press \([\text{ent}]\).
15. Touch the end of the part with the tool.
16. Move the tool away from the part, taking care not to move the Z axis.
17. Press the **Select Key** next to the ‘z’ and press \([\text{ent}]\) to zero the axis.
18. Repeat steps 10 to 17 for each tool to be set.
19. Press \([F2]\) to turn off the **Tool Set Mode**.
Using the tool offsets
1. Press [F2+] to turn on the Tool Usage Mode.
2. Press the left or right navigation key to select the tool. The function display shows the tool number ‘01’, ‘02’ etc., to ‘99’.
3. Press [F2+] to turn off the Tool Usage Mode.

Editing tool offsets for worn or replacement tools
1. Press [F2+] to turn on the Tool Usage Mode.
2. Press the left or right navigation key to select a known good tool.
3. Set the axis datum as described in steps 3 to 7 for Setting the tool offsets on page 9-3. All the offsets are now aligned with the correct Machine Datum.
4. Press [F2+] to turn off the Tool Usage Mode.
6. Set the offsets for each tool as described in steps 10 to 17 for Setting the tool offsets on page 9-3.
7. Press [F2] to turn off the Tool Set Mode.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.
To turn the function off, finish making any entry, then press the function key again.

Summing
You use this function to add the movement of the Z’ axis to the movement of either the X axis or the Z axis.
The Summing function is useful when the compound is set to align with either of those two axes. If the compound is set at an angle, see Vectoring on page 9-5.
Setting the parameters
You select the axes to be added together in Setup Mode.

To enter Setup Mode:
1. Leave any Special Function that is running
2. Press [set up].
   The centre display shows ‘set Up’.
3. Press the up or down navigation key to choose ‘add i t i o n’.
4. Press the left or right navigation key to choose which axes are to be added: X+Z’ or Z+Z’.
5. Press the up or down navigation key to choose ‘sto re’.
6. Press [ent] to store the change.

Using the summing function
Press [F1] or [F2] to turn the function on.
Press [F1] or [F2] to turn the function off.

For X + Z’
The X display shows the Sum of the two selected axes and the axis identifier shows ‘s’.
The Z display shows the Z axis as normal.
The Z’ display shows the Z’ axis as normal.

For Z + Z’
The X display shows the X as normal.
The Z display shows the Sum of the two selected axes and the axis identifier shows ‘s’.
The Z’ display shows the Z’ axis as normal.
You can zero or preset any of the axes in the usual way. The Sum display alters to take account of the new value. You may need to change the direction of Z’ in Setup to ensure that the axes sum and not subtract.

Vectoring
You use this function to combine the movement of the X and Z axes with the angle of the compound. Vectoring is only available on 3 axis units.

The Vectoring function is useful when the compound is set at an angle. If the compound is set to align with either the X or the Z axes, see Summing on page 9-4.
Using the vectoring function

1. Press [F1] or [F2] to turn the function on.
   The 'X' display shows Angle, and the centre display shows the Vectoring Angle.

2. Enter a new value if required.
   For example: Press [3] [5] [ent] to enter a vectoring angle of 35 degrees.

   The X display shows the combined X axis movement.
   The Z display shows the combined Z axis movement.
   The Z' display shows the Z' axis as normal.
   Press [F1] or [F2] to turn the function off.

You can zero or preset any of the axes in the usual way. The Vectoring displays alter to take account of the new value.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.

To turn the function off, finish making any entry, then press the function key again.

Combined X movement = \( X + Z'(\sin \alpha) \)
Combined Z movement = \( Z + Z'(\cos \alpha) \)
This chapter describes the special functions available in Generic mode.

In Generic mode all Lathe and Mill functions are also available.

The conventional way to set up a lathe is:
X Axis – cross travel
Z Axis – longitudinal travel
Z’ Axis – compound travel.

If you set the Machine Type to Mill or Generic, then the axes are labelled:
Axis 1 – X
Axis 2 – Y
Axis 3 – Z

Sub-datum
All Sub-Datums are relative to the Absolute Datum, so if you change the Absolute Datum, the Sub-Datums change accordingly.

The Sub-Datum function always works in Absolute Mode. If the C80 is in Incremental Mode when the Sub-Datum function is turned on, then it switches to Absolute Mode.

You can use other functions, such as Bolt Hole Circle in conjunction with Sub-Datums, to produce a repeated pattern of holes about different Sub-Datum positions.

With the Sub-Datum function you can store as many as 99 machining steps in the Sub-Datum Memory.

When you use the Sub-Datum function, the Absolute Datum of the machine is replaced by each Sub-Datum in turn, so you can work to zero for each step instead of having to constantly refer to a printed list of coordinates.

Example
SD = Sub-datum

---

Sub-datum memory

| SD1 | SD2 | SD3 | SD4 | SD5 | SD6 | SD7 | SD8 |...
|-----|-----|-----|-----|-----|-----|-----|-----|---
### Turning the function on and off

1. Press [F2] to turn the function on.
   The display shows:
   
   `sub dat`
   `job no.`

2. Press the **Select Key** next to the ‘s’.
   The display changes to ‘sd no.’.

3. Enter the number of the **Sub-Datum** that you want to go to.
   For example: Press [1] [ent] to go to **Sub-Datum 1**.
   The function display shows the **Sub-Datum** number ‘0 1’, ‘0 2’ up to ‘99’.

4. Press the left or right navigation key to step from one **Sub-Datum** to the next.

5. Press [abs/inc] to turn the function off.

### Setting a sub-datum

Go to the **Sub-Datum** that is to be set, then use either of these two methods:

#### Teach method

1. Move the machine to the position to be stored as the **Sub-Datum**.

2. Press [F2+].
   All displays show ‘0 0 0 0’.

   This sets the **Sub-Datum**.

#### Preset method

You do not need to move the machine.

1. Press [preset].

2. Press the **Select Key** next to the first axis that you want to set.

3. Enter the position of the **Sub-Datum** relative to the absolute datum.

4. Press [F2].
   The display shows the distance from the current machine position to the **Sub-Datum**.

5. Set any other axes that need to be set.

### Inserting a sub-datum

1. Go to the point where the new **Sub-Datum** is to be inserted.

2. Press [ins].
   The display shows:
   ‘i ns sub’
   ‘i ns job’.

3. Press the **Select Key** next to the ‘s’.

   *Pressing any other key cancels the operation.*
Jobs

After a short time delay, all the following **Sub-Datum** numbers are incremented by one, and the display shows the current machine position.

4. Set the new **Sub-Datum** as described in Setting a sub-datum on page 10-2.

**Deleting a sub-datum**

1. Go to the Sub-Datum that you want to delete.

2. Press [del].
   The display shows ‘del sub’.

3. Press the Select Key next to the ‘s’.
   *Pressing any other key cancels the operation.*

After a short time delay all the following **Sub-Datum** numbers are decremented by one, and the display shows the next **Sub Datum**.

**Jobs**

You use this function to divide the stored **Sub-Datums** into groups so that you can identify a number of individual **Jobs**.

When you insert **Job** markers all the following **Sub-Datum** numbers are incremented to account for the presence of the markers in memory. This is because of the way in which the **Sub-Datum Memory** is used.

**Example**

- **Jobs**
  - **Start**
  - **End**
  - **Job 1**
  - **Job 2**
- **Sub-Datum Memory**
  - **SD2**
  - **SD3**
  - **SD4**
  - **SD5**
  - **SD6**
  - **SD9**
  - **SD10**
  - **SD11**

**Inserting a job marker**

1. Go to the point where you want to insert the new **Job** marker.

2. Press [ins].
   The display shows:
   ‘ins sub’
   ‘ins job’.

3. Press the Select Key next to the ‘j’.
   *Pressing any other key cancels the operation.*
The display changes to:
‘i n s e r t ’
‘j o b ’.

**For a start job marker**

1. Press [ent] to confirm.
   The display changes to ‘j o b n o .’.
2. Enter the number of the Job that you want to add.

For example: Press [1] [ent] to insert the marker for Start Job 1.

All the following Sub-Datum numbers are incremented by one, and the display shows the new Start Job marker.

**For an end job marker**

1. Press the up or down navigation key.
   The display changes to ‘e n d j o b ’.
2. Press [ent] to confirm.

All the following Sub-Datum numbers are incremented by one, and the display shows the new End Job marker.

**Deleting a job marker**

1. Go to the Job marker that you want to delete.
2. Press [del].
   The display shows ‘d e l s u b ’.
3. Press the Select Key next to the ‘s ’.

*Pressing any other key cancels the operation.*

All the following Sub-Datum numbers are decremented by one, and the display shows the next Sub-Datum.

**Finding a job**

1. Press [F2].
   The display shows:
   ‘s u b d a t ’
   ‘j o b n o .’
2. Press the Select Key next to the ‘j ’.
   The display changes to ‘j o b n o .’.
3. Enter the number of the Job that you want to find.
   For example: Press [2] [ent] to find Job 2.
   If you enter an invalid Job number, the display shows Job 1.
4. Press the up or down navigation key to choose a valid job number.
5. Press [ent] to continue.

The display shows the Start Job marker.

Once the function is on, you can go from one Sub-Datum to another, by either of these two methods:
Press [F2], then the Select Key by the ‘s ’. Enter the number of the Sub-Datum that you want to go to.
Press the left or right navigation key to step from one **Sub-Datum** to the next.

If you make a mistake while entering a number, you can press [ce] to clear the entry one character at a time.

To find the first **Sub-Datum** of the **Job**, press the right navigation key.

To turn the function off, finish making any entry, then press the function key again.
This chapter describes some of the errors that might occur, and ways that you might be able to solve them.

<table>
<thead>
<tr>
<th>Symptom</th>
<th>Solution</th>
</tr>
</thead>
</table>
| The display is blank.                                                   | • The **C80** may be in **Sleep Mode**. Press the **Sleep Mode** key.  
  • Check that the power supply is correctly connected to a working mains outlet.  
  • Check that the power supply cables are not damaged.  
  • Check that the power supply voltage is 15Vdc ±10%.  
  • Disconnect all encoder cables. A defective encoder can prevent the **C80** from working.  
  • Check the power supply indicator on the rear of the **C80** display to ensure that it is illuminated. |
| The display works, but resets from time to time without any keys being pressed. | Either the supply voltage is too low, or the power supply or mains supply has an intermittent fault.  
  • Check that the power supply voltage is 15Vdc ±10%.  
  • Check that all the connections are secure. |
| The display works, but gives erratic readings, the last digit jitters or the measurements jump to new figures unexpectedly. | There may be a poor earth (ground) connection. Both the **C80**, and the machine on which it is installed, must have proper earth (ground) connections (see Cable connections on page 2-1).  
There may be a problem with the encoder. |
| The unit does not respond to any key presses.                          | Disconnect the **C80** from its power supply, wait 15 seconds and then reconnect.                                                                                                                             |
## Troubleshooting

<table>
<thead>
<tr>
<th>Symptom</th>
<th>Solution</th>
</tr>
</thead>
</table>
| 'no Sig' or 'SIG FAIL' appears in the display. | This indicates that the unit is not receiving a proper signal from the encoder.  
- Check that the encoder connections are secure.  
- Check that there is no damage to the connectors or to the encoder.  
- Switch the C80 off and back on again.  
- Swap the encoder to another axis to confirm whether the encoder or the C80 is at fault. See To swap encoders to trace a fault: on page 11-3. |
| Readings are incorrect. | - Check the Encoder Type to ensure it is correct.  
- Check the Radius / Diameter setting. The Diameter setting causes the axis to read double.  
- Check the Error Compensation factors.  
- If using the Segmented Error Compensation, verify the datum position.  
- Swap the encoder to another axis to confirm whether the encoder or the C80 is at fault. See To swap encoders to trace a fault: on page 11-3.  
- Check that there is no damage to the encoder or its cable.  
- Check that the encoder is fixed firmly and aligned correctly, as described in the Spherosyn / Microsyn Installation manual.  
- Check that there is no binding on the scale. With the scale brackets slightly loosened, you should be able to slide the scale back and forth with minimal resistance.  
- If you have a Spherosyn scale, check that the scale is not bent, by removing it and rolling it on a flat surface. |
If the solutions suggested above do not solve your problem, contact Newall for further instruction.

To swap encoders to trace a fault:

1. Check that the two axes are set to the correct encoder types.
2. Disconnect the C80 power supply.
3. Move the encoder from the malfunctioning axis to a working axis.
4. Reconnect the C80 power supply and turn on.

If the fault stays with the same encoder, then the encoder is at fault. If the fault does not follow with the encoder the C80 is at fault.

Providing you have not moved the machine more than 6.3mm (0.25”) for a Spherosyn Encoder or 2.5mm (0.1”) for a Microsyn Encoder, switching the power off and back on again does not lose the datum position.
This chapter describes how to clean your C80 without damaging it.

You should follow these instructions carefully to avoid damaging the C80.

To clean your C80:

1. Disconnect the power supply from the C80.

2. Apply a small amount of mild soap to a lint-free cloth. Use this to wipe over the case and keypad, taking care not to allow fluid into the connectors.

Do not use corrosive or abrasive cleaning materials.

Do not use compressed air.
Index

A
Abbé error 5-5
Absolute datum 5-6, 6-1, 10-1
Absolute mode 4-1, 6-1, 6-2, 9-2
Arc centre 8-3
Arc radius 8-3
Axis displays 4-1
Axis ID 5-10
Axis summing 5-12
B
Bolt hole circle 8-1
C
Calculating the correction factor 5-8
Cartesian coordinate system 8-7
centre 6-1
Circle centre 8-1
Circle diameter 8-1
Clear entry 6-1
Continuous output 5-9
Correction factor 5-8, 5-9
Correction parameters 5-6
Correction point 5-6, 5-7
D
Datum 6-1
Defaults 5-1
del 10-3, 10-4
Deleting a job marker 10-4
Deleting a sub-datum 10-3
Depth 1-1
Diameter 5-3
Direction 5-3
Display
  blank 11-1
  erratic readings 11-1
  no sig 11-2
  resets unexpectedly 11-1
  sig fail 11-2
Disposal 1-2
DRO 5-3
E
EMC compliance 1-1
Encoder 5-2
Encoder resolution 5-2
Encoder type 5-2
End job marker 10-4
Ending point 8-3
Environmental conditions 1-1
Error compensation 5-3
External cut 8-3
F
Feedrate 4-1
Finding a job 10-4
Function display 4-1
Function keys 4-2
G
Generic mode 7-1
H
Height 1-1
I
Identifying the correction parameters 5-6
in/mm 5-2
Incorrect readings 11-2
Incremental mode 4-1, 6-2, 9-1, 9-2
ins 10-2, 10-3
Inserting a job marker 10-3
Inserting a sub-datum 10-2
Internal cut 8-3
J
Jobs 10-3
K
Keyed output 5-9
Keypad 4-1
Known fixed value 6-1
L
Lathe 4-1
Lathe mode 7-1
Leaving setup mode 5-2
Line angle 8-5
Line length 8-5
Linear error compensation 5-8
Low voltage compliance 1-1
M
Machine datum 9-2, 9-3
Maximum cut 8-3
menu 7-2
Microsyn 6-1
Microsyn 10 resolution 1-2
Microsyn 5 resolution 1-2
Mill 4-1
Mill mode 7-1
Mode key 4-1
N
Navigation and function keys 4-2
Navigation keys 4-2
Normal operating mode 5-9
Number of holes 8-1, 8-5
O
Operating temperature 1-1
Index

P
PCD 8-1
Pitch circle diameter 8-1
Pitch error 5-5
Plane 8-1, 8-3, 8-5
Point-to-point 4-1
Polar coordinate system 8-7
Power supply 1-1
preset 6-1
Preset method 10-2
Presetting an axis 6-1

R
Radius 5-3
recall 6-1
Recalling the last value 6-1
ref 6-1
Reference point 5-6, 5-7
Relative humidity 1-1
Resolution 5-2
Roll error 5-5

RS232
connections 5-10
default system settings 5-10
operation 5-11
options 5-9
output data format 5-10
system settings 5-10

S
Segmented compensation 5-7
Segmented error compensation 5-5
set up 9-1
Setting a sub-datum 10-2
Setting the correction factor 5-9
Setting the correction points 5-6
Setting the preset 6-1
Setting the zero 6-1
Setup 4-1
Sleep key 4-2
Sleep mode 4-2, 11-1
Special functions 4-1
Spherosyn 6-1
Spherosyn resolution 1-2
Start job marker 10-4
Starting angle 8-1
Starting point 5-6, 5-7, 8-3, 8-5
Index

Storage temperature 1-1
Sub-datum memory 10-1

T
Taper angle 9-1
Taper display axis 5-12
Teach method 10-2
Temperature
  operating 1-1
  storage 1-1
Tool 1 offset 9-2
Tool diameter 8-3
Tool offsets datum 9-3
Tool set mode 9-2
Tool usage mode 9-3

U
Unit key 4-1
Using Centerfind 6-2
Using Digifind 6-1

W
Way errors 5-5
Weight 1-1
Width 1-1

X
X, Y and Z axes 4-1

Y
Yaw error 5-5

Z
Zero an axis 6-1
Zero approach 5-12
Zero approach limit 5-12
Zero approach off 5-12
Zero approach on 5-12