ACU-RITE SOLUTIONS

droPWR

Operating Instructions

iOS application

Contents

Contents

1

Fundamentals

1.1 About these instructions

These instructions provide all the information and safety precautions needed for the safe operation of the product.

1.2 Notes on reading the documentation

The table below lists the components of the documentation in the order of priority for reading.

> ⚠️ **WARNING**
>
> **Fatal accidents, personal injury or property damage caused by non-compliance with the documentation!**
>
> Failure to comply with the documentation may result in fatal accidents, personal injury or property damage.
>
> ▶ Read the documentation carefully from beginning to end
> ▶ Keep the documentation for future reference

Documentation	Description
Addendum	An addendum supplements or supersedes the corresponding contents of the Operating Instructions and, if applicable, of the Installation Instructions. If this document is included in the shipment, read it first before you proceed. All other contents of the documentation retain their validity.
Installation Instructions	The Installation Instructions contain all the information and safety precautions needed for the proper mounting and installation of the product. They are an excerpt from the Operating Instructions and are included in every shipment. This document has the second highest priority for reading.
Operating Instructions	The Operating Instructions contain all the information and safety precautions needed for the proper operation of the product according to its intended use. This document has the third highest priority for reading. This documentation can be downloaded from the download area at **www.acu-ritesolutions.com**. The Operating Instructions must be printed prior to commissioning the product.
Documentation of connected measuring devices and other peripherals	These documents are not included in delivery. They are shipped with the respective measuring devices and peripherals.

Would you like any changes, or have you found any errors?

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address:

acu-ritesolutions@heidenhain.com

1.3 Storage and distribution of the documentation

The instructions must be kept in the immediate vicinity of the workplace and must be available to all personnel at all times. The operating company must inform the personnel where these instructions are kept. If the instructions have become illegible, the operating company must obtain a replacement from the manufacturer.

If the product is given or resold to any other party, the following documents must be passed on to the new owner:

- Addendum, if supplied
- Installation Instructions
- Operating Instructions

1.4 Target group for the instructions

These instructions must be read and observed by every person who performs any of the following tasks:

- Mounting
- Installation
- Commissioning
- Setup, programming and operation
- Service, cleaning and maintenance
- Troubleshooting
- Removal and disposal

1.5 Notes in this documentation

Safety precautions

Comply with all safety precautions indicated in these instructions and in your machine tool builder's documentation!

Precautionary statements warn of hazards in handling the product and provide information on their prevention. Precautionary statements are classified by hazard severity and divided into the following groups:

⚠ DANGER
Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard will **result in death or severe injury.**

⚠ WARNING
Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury.**

⚠ CAUTION
Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in things other than personal injury, like property damage**.

Informational notes

Observe the informational notes provided in these instructions to ensure reliable and efficient operation of the product.
In these instructions, you will find the following informational notes:

> **i** The information symbol indicates a **tip**.
>
> A tip provides important additional or supplementary information.

> **⚙** The gear symbol indicates that the function described **depends on the machine,** e.g.
>
> ■ Your machine must feature a certain software or hardware option
>
> ■ The behavior of the function depends on the configurable machine settings

> **📖** The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine tool builder or other supplier.

1.6 Symbols and fonts used for marking text

In these instructions the following symbols and fonts are used for marking text:

Format	Meaning
▶ ... > ...	Identifies an action and the result of this action Example: ▶ Tap the **Save & Set** button > The datum is saved and set as the current datum
■ ... ■ ...	Identifies an item of a list Example: ■ **Milling** ■ **Turning**
Bold	Identifies menus, screens, displays, and buttons Example: ▶ Tap the **Set Tool** button > The tool table is displayed.

2

Safety

2.1   Overview

This chapter provides important safety information needed for the proper mounting, installation and operation of the product.

2.2   General safety precautions

General accepted safety precautions, in particular the applicable precautions relating to the handling of live electrical equipment, must be followed when operating the system. Failure to observe these safety precautions may result in personal injury or damage to the product.

It is understood that safety rules within individual companies vary. If a conflict exists between the material contained in these instructions and the rules of a company using this system, the more stringent rules take precedence.

2.3   Intended use

> **i**   Knowledge of iPad navigation and use is required when using droPWR.

Acu-Rite Solutions droPWR is a digital readout application for iPad. The droPWR app displays the position of the tool and provides functions for operating the machine tool when combined with an Acu-Rite Solutions IBT400, a manually operated machine tool, and linear or angle encoders.

Minimum system requirements

iPad	iPadOS 15

> 📖   The droPWR application supports the use of a wide variety of peripheral devices from different manufacturers. HEIDENHAIN cannot make any statements on the intended use of these devices. The information on their intended use, which is provided in the associated documentations, must be observed.

2.4   Improper use

When the product is used it must be ensured that no hazard to persons can result. If any such hazard exists, appropriate measures must be taken by the operating company.

In particular, the product must not be used in the following applications:

- Use and storage outside the specifications
- Outdoor use
- Use in potentially explosive atmospheres
- Use of the product as part of a safety function

2.5 Personnel qualification

The personnel for mounting, installation, operation, service, maintenance and removal must be appropriately qualified for this work and must have obtained sufficient information from the documentation supplied with the product and with the connected peripherals.

The personnel required for the individual activities to be performed on the product are indicated in the respective sections of these instructions.

The personnel groups that are responsible for mounting, installation, operation, maintenance and removal have different qualifications and tasks, which are specified as follows.

Operator

The operator uses and operates the product within the framework specified for the intended use. He is informed by the operating company about the special tasks and the potential hazards resulting from incorrect behavior.

Qualified personnel

The qualified personnel are trained by the operating company to perform advanced operation and parameterization. The qualified personnel have the required technical training, knowledge and experience and know the applicable regulations, and are thus capable of performing the assigned work regarding the application concerned and of proactively identifying and avoiding potential risks.

Electrical specialist

The electrical specialist has the required technical training, knowledge and experience and knows the applicable standards and regulations, and is thus capable of performing work on electrical systems and of proactively identifying and avoiding potential risks. Electrical specialists have been specially trained for the environment they work in.

Electrical specialists must comply with the provisions of the applicable legal regulations on accident prevention.

2.6 Obligations of the operating company

The operating company owns or leases the product and the peripherals. It is responsible that the intended use is complied with at all times.

The operating company must:

- Assign the different tasks to be performed on the product to appropriate, qualified, and authorized personnel
- Verifiably train personnel in the tasks they are allowed to perform and how to perform them
- Provide all materials and means necessary in order for personnel to complete the assigned tasks
- Ensure that the product is operated only when in perfect technical condition
- Ensure that the product is protected from unauthorized use

2.7 General safety precautions

> **ⓘ** The safety of any system incorporating the use of this product is the responsibility of the assembler or installer of the system.

> **📖** The product supports the use of a wide variety of peripheral devices from different manufacturers. HEIDENHAIN cannot make any statements on the specific safety precautions to be taken for these devices. The safety precautions provided in the respective documentations must be observed. If no such information has been supplied, it must be obtained from the manufacturers concerned.

The specific safety precautions required for the individual activities to be performed on the product are indicated in the respective sections of these instructions.

2.7.1 Symbols in the instructions

The following safety symbols are used in this manual:

Symbol	Meaning
⚠️	Identifies information that warns of personal injury
📛	Identifies electrostatic sensitive devices (ESD)
✋	ESD wristband for personal grounding

3

Fundamentals of positioning

3.1   Overview

This chapter describes basic positioning information.

3.2   Datums

The workpiece drawing identifies a certain point on the workpiece (example: "a corner") as the absolute datum, and perhaps one, or more other points as relative datums.

The datum setting procedure establishes these points as the origin of the absolute, or relative coordinate systems. The workpiece, which is aligned with the machine axes, is moved to a certain position relative to the tool. The display is set to zero.

3.3   Actual position, nominal position, and Distance-To-Go
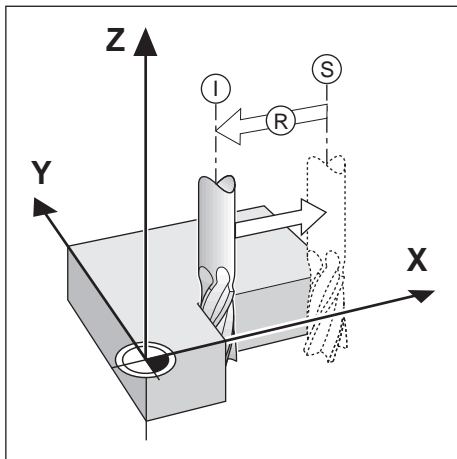
The position of the tool at any given moment is called the Actual Position **I**, while the position that the tool is to move to is called the Nominal Position **S**. The distance from the nominal position to the actual position is called the Distance-To-Go **R**.

3.4 Absolute workpiece positions

Each position on the workpiece is uniquely identified by its absolute coordinates.
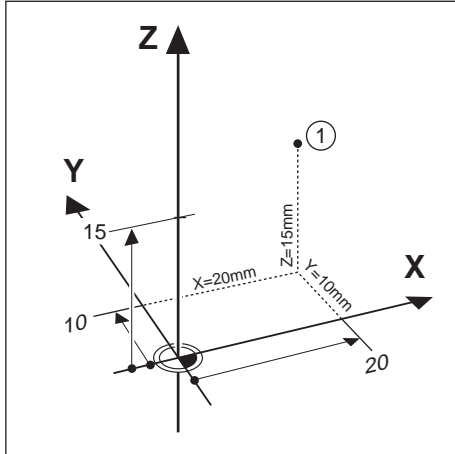
Example: Absolute coordinates of position **1**:
X = 20 mm
Y = 10 mm
Z = 15 mm

If you are drilling, or milling a workpiece according to a workpiece drawing with absolute coordinates, the tool is moving to the value of the coordinates.

3.5   Incremental workpiece positions

A position can also be referenced to the preceding nominal position. In this case the relative datum is always the last nominal position. Such coordinates are referred to as Incremental Coordinates. They are also called incremental, or chain dimensions, since the positions are defined as a chain of dimensions. Incremental coordinates are designated with the prefix **I**.
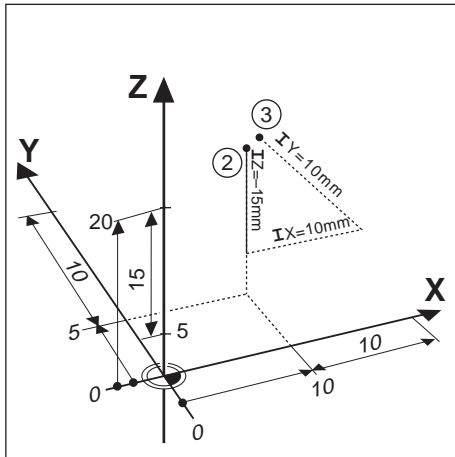
Example: Incremental coordinates of position **3** referenced to position **2**.

Absolute coordinates of position **2**:

X = 10 mm
Y = 5 mm
Z = 20 mm

Incremental coordinates of position **3**:

IX = 10 mm
IY = 10 mm
IZ = 15 mm

If you are drilling, or milling a workpiece according to a drawing with incremental coordinates, you are moving the tool by the value of the coordinates.

3.6 Zero angle reference axis
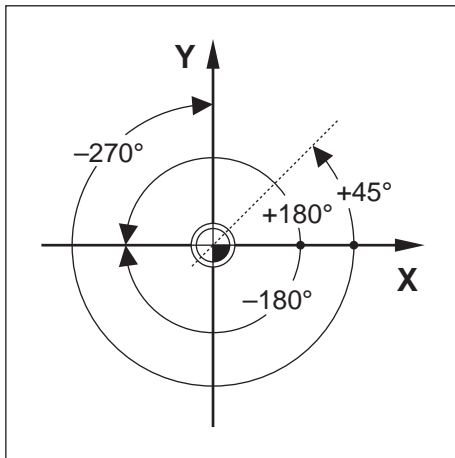
The Zero Angle Reference Axis is the 0.0° position. It is defined as one of the two axes in the plane of rotation. The following table defines the Zero Angle where the position of the angle is zero for the three possible planes of rotation.

For angular positions, the following reference axes are defined:

Plane	Zero Angle Reference Axis
XY	+X
YZ	+Y
ZX	+Z

Positive direction of rotation is counterclockwise if the working plane is viewed in the negative tool axis direction.

Example: Angle in the working plane X / Y

Plane	Zero Angle Reference Axis
+45°	... bisecting line between +X and +Y
+/-180°	... negative X axis
-270°	... positive Y axis

3.7 Reading head position
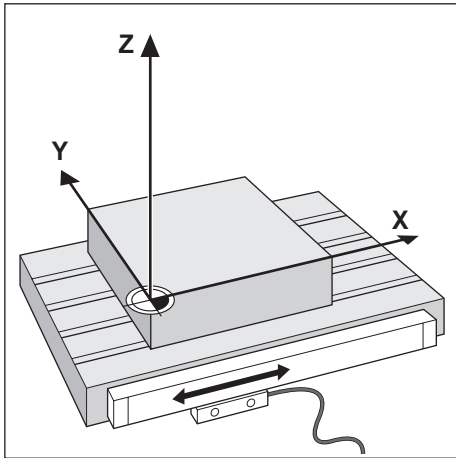
The reading head position provides feedback to the product that converts the movement of the machine axes into electrical signals. The product constantly evaluates these signals, calculates the actual positions of the machine axes, and displays the positions as a numerical value on the display.

If there is an interruption in power, the calculated position will no longer correspond to the actual position. When power is restored, you can re-establish this relationship by using the reference marks on the encoder. This product provides the Reference Mark Evaluation Feature (REF).

3.8   Encoder reference marks

Encoders normally contain one or more reference marks which the Reference Mark Evaluation feature uses to re-establish datum positions after a power interruption. There are two main options available for reference marks:

- Fixed reference marks
- Distance-coded reference marks

Fixed reference marks
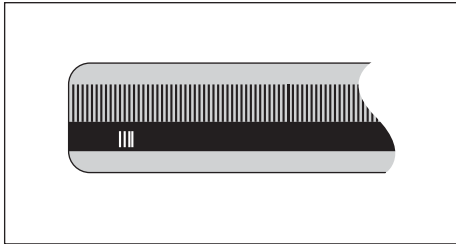
Encoders that have one or more marks on fixed intervals have to re-establish the datums correctly. It is necessary to use the same exact reference mark, during the Reference Mark Evaluation routine, that was used when the datum was first established.

Position Trac (Distance-coded reference marks)
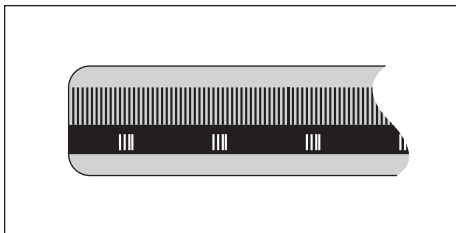
Encoders that have marks separated by a specific encryption pattern allow the product to use any two pair of marks along the length of the encoder to re-establish the prior datums. This configuration means that you only have to travel less than 20 mm anywhere along the encoder to re-establish the datums when the product is switched back on.

> ℹ️ The established datums cannot be restored from one power cycle to the next if the reference marks were not crossed before the datums were set.

4

Basic operation

4.1 Overview

This chapter describes the product's user interface and basic functions.

4.2 User interface

4.2.1 Display layout
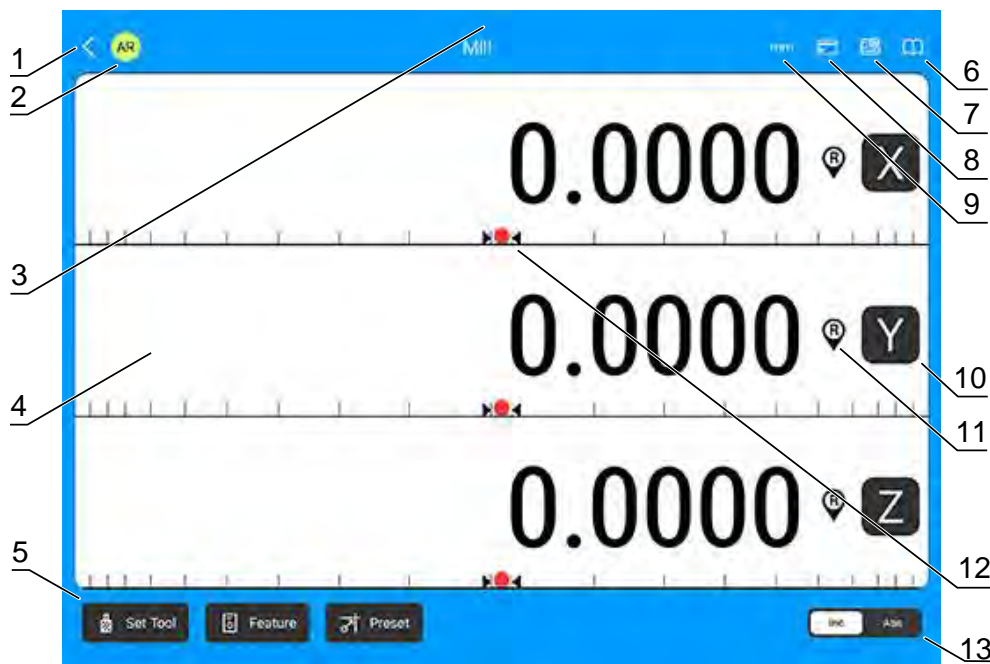
1 Settings
2 Avatar
3 Split view
4 Display area
5 Functions
6 Books
7 View settings
8 Subscription
9 Unit of measure
10 Axis labels
11 Reference mark indicator
12 Graphic positioning aid
13 Operating mode
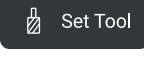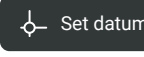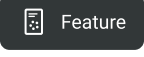
Feature	Function
Settings	Provides access to the application settings menus
Avatar	Indicates the current user
Split view	Provides access to Split View
Display area	Indicates the current position of each axis. Also shows forms, fields, instruction boxes, error messages, and help topics
Functions	Indicate various available functions based on the current operating mode or menu
Books	Provides access to the Books app

Feature	Function
View Settings	Provides access to the **View Settings** menu
Subscription	Provides access to the droPWR subscription options
Unit of measure	Indicates current unit of measure
Axis labels	Indicates axis and provides access to axis menu
Reference mark indicator	Indicates current reference mark status ⊙ Reference marks are established. A flashing indicator represents that reference mark detection is enabled, but the reference marks have not yet been established. ⊘ Reference marks are not established
Graphic positioning aid	Indicates the Distance-To-Go
Operating mode	Indicates the current operating mode and allows you to toggle between Incremental and Absolute mode.

4.2.2   Operating elements

The following operating elements are used in the application:

Operating element	Function
<	**<** Tap **<** to access **Machine Tools**, **My Profile**, and **App Settings**
Inch / mm	**inch**/**mm** Tap **inch** or **mm** to toggle the unit of measure
[subscription icon]	**Subscription** Tap the **Subscription** icon to access the droPWR subscription options
[view settings icon]	**View Settings** Tap the **View Settings** button to access settings for adjusting the readout layout
[books icon]	**Books** Tap the **Books** icon to open the Books app to access the droPWR Operating Instructions
Set Tool	**Set Tool** Tap the **Set Tool** button to open the Tool Table
Set datum	**Set datum** Tap the **Set datum** button to open the Datum form and set the datum for each axis. Available in **Abs** mode only.
Preset	**Preset** Tap the **Preset** button to open the Preset form. This form is used to set a nominal position. This is a Distance-To-Go function. Available in **Inc** mode only.
Feature	**Feature** Tap the **Feature** button to create **Circle Patterns** or **Linear Patterns**
Inc Abs	**Inc/Abs** Tap **Inc**/**Abs** to toggle between Distance-To-Go **Inc** mode and Actual Value **Abs** mode
[enable reference mark icon]	**Enable Reference Mark** Tap the Enable Reference Mark button when you are ready to identify a reference mark
[disable reference mark icon]	**Disable Reference Mark** Tap the Disable Reference Mark button when you want to cross over a reference mark and have the system ignore the reference mark or to exit the reference mark evaluation routine and work without reference marks
[teach icon]	**Teach** Tap the **Teach** button to insert the current position into the selected field

Operating element	Function
	Trash Tap the **Trash** button to delete the selected feature
1/2	**1/2** Tap **1/2** to divide the current position by two. Available in Mill application only.
Rad Dia	**Rad/Dia** Tap **Rad/Dia** to toggle between diameter and radius measurements. Available in Turn application only.
ZERO	**zero** Tap the **zero** button to zero an axis

4.2.3 Graphic positioning aid

When positioning to the next nominal position, the product assists you by displaying the Distance-To-Go as well as a graphic positioning aid (traversing to zero). A scale is shown underneath each axis you traverse to zero. The graphic positioning aid is a red circle that symbolizes the axis slide.

The positioning aid moves across the scale when the axis slide is within ± 5 mm of the nominal position.

4.2.4 Operating modes

The product has two operating modes:
- Distance-To-Go (Incremental)
- Actual Value (Absolute)

Distance-To-Go mode (Incremental)

The Distance-To-Go mode enables you to approach nominal positions by zeroing the axes and traversing to a position based on its distance from the zeroed position.
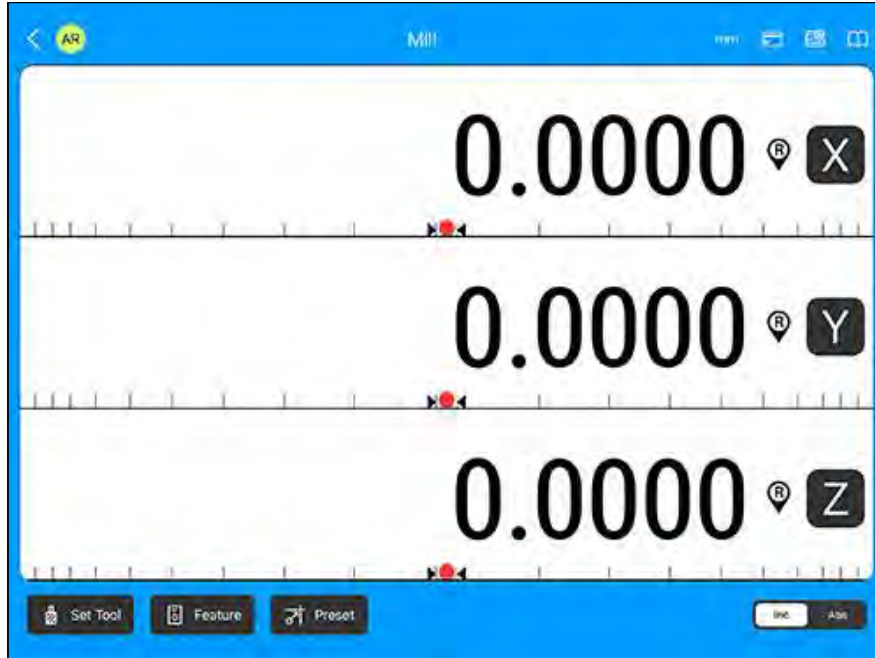
Actual Value mode (Absolute)

The Actual Value mode always displays the current actual position of the tool, relative to the active datum. In this mode, all moves are done by traveling until the display matches the nominal position that is required.
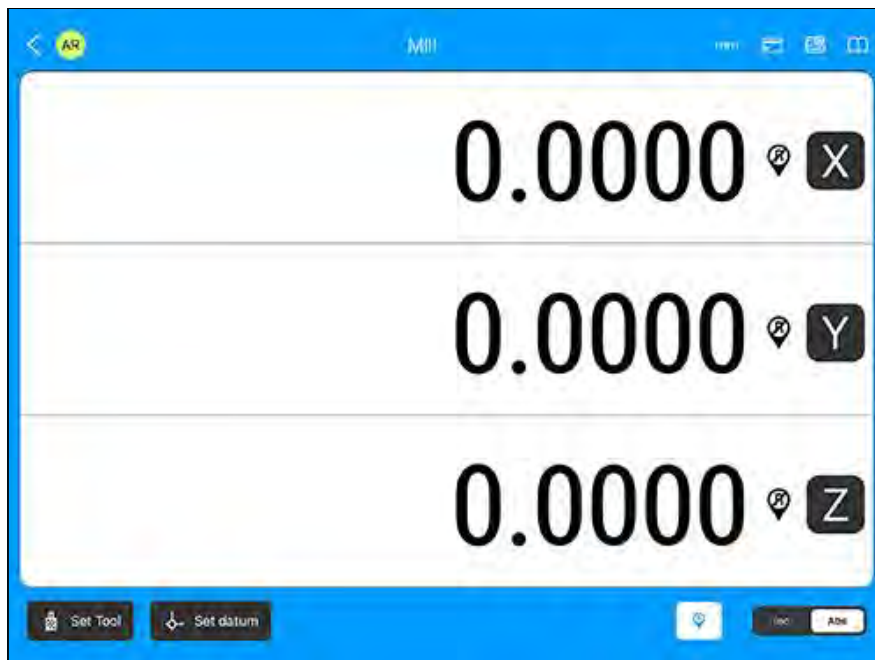
Changing the operating mode

To change the operating mode:

▶ Tap **Inc** or **Abs** in the **Inc**/**Abs** toggle to switch between operating modes

4.2.5 Data input forms

Information required for various operational functions and setup parameters are entered through data input forms. These forms will appear after selecting features that require additional information. Each form provides specific fields for entering the required information.

4.2.6 Reference mark evaluation

The Reference Mark Evaluation feature automatically re-establishes the relationship between axis slide positions and display values that was last defined by setting the datum.

The reference mark indicator will flash for each axis with an encoder that has reference marks. The indicator will stop flashing after crossing over the reference marks.

Activating reference marks
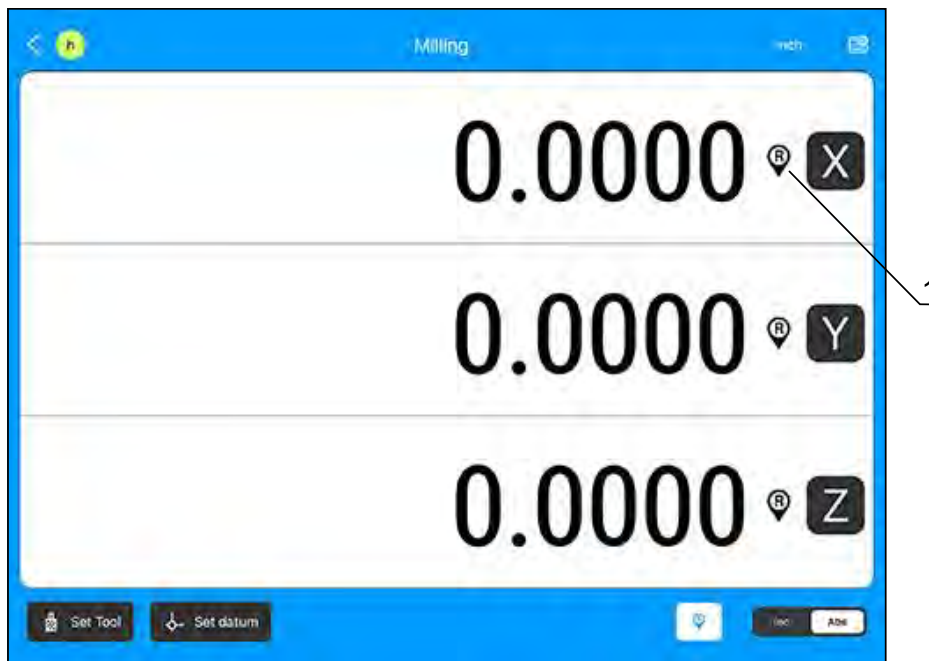
To activate reference marks:

▶ Cross over the reference marks for each axis to activate the reference

> After a successful reference mark evaluation, the indicator will stop flashing

Working without reference marks

The product can also be used without crossing over the reference marks.
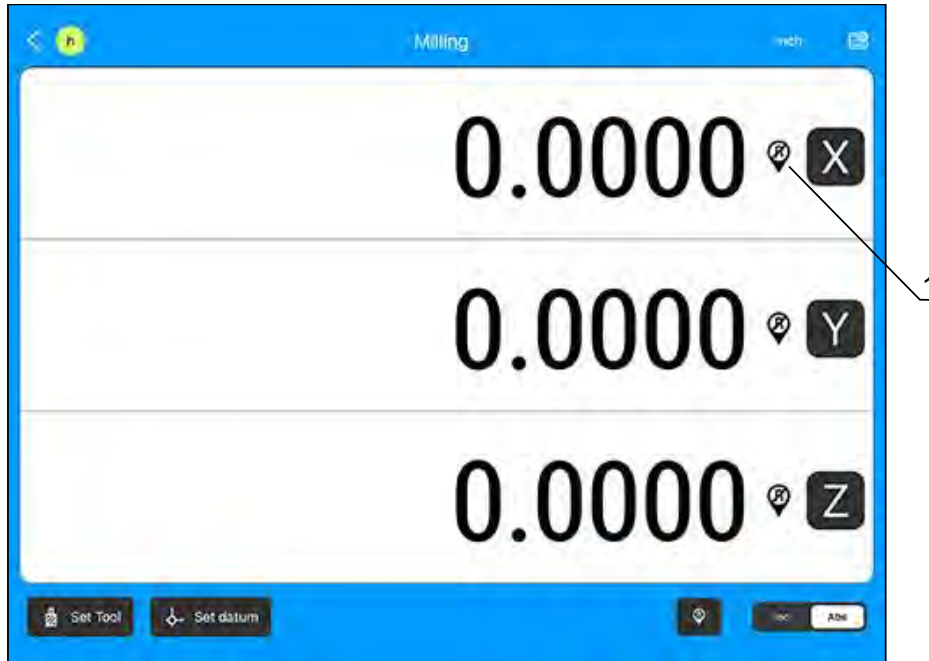
	To disable reference marks:
⊙	▶ Tap the reference mark active button to exit the reference mark evaluation routine, and continue
	> After disabling reference marks the indicator will have a forward slash through it, signifying that the reference marks have been disabled

Re-enabling reference marks

Reference marks can be enabled any time after they have been disabled

	To re-enable reference marks:
⊙	▶ Tap to the reference marks not active button to activate the reference mark evaluation routine

---

🛈   If an encoder is setup without reference marks the reference indicator will not be displayed. A datum set from the axis will be lost once the application is closed.

4.2.7   Selecting a specific reference mark

The product provides the ability to select a specific reference mark on an encoder. This is important when using encoders with fixed reference marks.

To select a specific reference mark:
▶ Tap the active reference marks button to deactivate reference marks
> The button will turn black. The evaluation routine is paused. Any reference marks that are crossed during encoder movement are ignored.
▶ Cross over undesired reference marks

▶ Tap the inactive reference marks button to reactivate reference marks
> The button will turn white. The next reference mark that is crossed will be selected.
▶ Cross the desired reference mark
▶ Repeat this process for all desired reference marks

▶ Tap the active reference marks button to cancel out of the routine after all desired axes are established
> Only the axes that are needed require crossing over the reference marks. If all reference marks have been found the product will return to the DRO display screen automatically.

4.2.8   Coupling

The application provides a quick method for coupling axes inputs.

Moving either of the coupled inputs will update the position on the coupled display.

Coupling is preserved in between power cycles.

> ⓘ   The reference mark for both encoders must be found to recall the previous datum when inputs are coupled.

To couple inputs:
▶ Tap **<**
▶ Tap **Machine Tools**
▶ Tap the machine tool with the inputs to couple
▶ Tap **Displays**
▶ Select the display where you would like to show the coupled axes
▶ Tap the **Couple Operation** pull-down
▶ Select **+** or **-**
▶ Tap the **Input Two** pull-down
▶ Select the second display
▶ Tap the **Save** button
> The sum of the coupled inputs will be shown on the coupled display

4.2.9 Error messages

If an error occurs while working with the product, a message will appear on the display, and provide an explanation of what caused the error.

To clear an error message:

▶ Tap the **Clear** button

> The error message will be cleared and normal operation can continue

5

Commissioning

5.1   Overview

> **ℹ**   Make sure that you have read and understood the "Basic Operation" chapter before performing the activities described in this section.
> **Further information:** "Basic operation", Page 23

> **ℹ**   The following steps are only to be performed by qualified personnel.
> **Further information:** "Personnel qualification", Page 13

Backing up the configuration

Configuration data can be backed up after commissioning. The configuration data can be reused for equivalent products.

Further information: "App Settings", Page 43

5.2   Setup guide

A setup guide is provided the first time you open the application. This guide will step you through common commissioning parameters.

> **ℹ**   ***** An asterisk indicates a required field.

5.2.1   Allow Bluetooth

To allow Bluetooth use:

▶ Tap **OK** to allow droPWR to use Bluetooth

> **ℹ**   Bluetooth must be enabled in iPad settings for droPWR to communicate via Bluetooth.

5.2.2   Agree to Terms of Use

To agree to the terms of use:

▶ Read the Terms of Use
▶ Tap the checkbox to agree to the terms of use
▶ Tap the **Start Setup** button to begin setup

5.2.3   Select a subscription

To select a subscription:

▶ Tap a radio button next to the subscription you would like to select
  ▪ IBT Devices Starter
  ▪ Demo mode (free)
  Further information: "Subscription", Page 45
▶ Tap **Continue**
▶ Tap **Subscribe**
▶ Tap **OK**

5.2.4 Create a supervisor user

To create a supervisor user:
▶ Create a **Screen name**
▶ Create a **PIN code ***
▶ Enter your **Email address ***
▶ Tap the **Save user** button to save the the supervisor user
▶ Tap the **Step 2** button to continue and begin pairing a machine tool

5.2.5 Pair a machine tool

To pair a machine tool:
▶ Select the desired bluetooth enabled machine tool from the **Available bluetooth devices**

> **ⓘ** IBT devices are identified by the device name followed by the last four digits of the serial number.
> ex. **IBT400 1234**

▶ Tap the **Scan Again** button if the required device is not listed in the **Available bluetooth devices**
Once connected to the machine tool:
▶ Tap the **Step 3** button to continue and begin the machine tool setup

5.2.6 Setup the machine tool

To setup the machine tool:
▶ Select the application
   ▪ **Mill**
   ▪ **Turn**
▶ Enter a **Machine tool name ***
▶ Select the **Number of Encoders**
▶ Select the **Default unit** of measure
   ▪ **inch**
   ▪ **mm**
▶ Tap the **Save & Continue** button
▶ Tap the **Step 4** button to continue and configure the encoders

5.2.7   Configure the encoders

To configure the encoders:

▶ Select the **Encoder Type**

- ■ **Linear**
- ■ **Rotary**
- ■ **Linear (rotary)**

▶ Select the **Encoder Resolution (µm)** or enter using the keypad

▶ Select the **Reference mark** type

- ■ **Single**
- ■ **CODED/2000**
- ■ **EverTrack**
- ■ **P-trac**
- ■ **Position trac**
- ■ **None**

▶ Select the **Count Direction**

- ■ **Positive**
- ■ **Negative**

▶ Tap the toggle to enable or disable counting **Error monitor**

▶ Tap the **Next** button

▶ Repeat the previous steps to configure each encoder

▶ Tap the **Save & Continue** button

▶ Tap the **Step 5** button to continue and configure the encoder displays

5.2.8   Setup the Displays

To setup the displays:

X   ▶ Tap the axis button and select an axis label

■ X	■ Y	■ Z	■ U
■ V	■ W	■ A	■ B
■ C	■ S	■ T	■ Q

▶ Select the **Display Resolution**

■ 0.05	■ 0.02	■ 0.01	■ 0.005
■ 0.002	■ 0.001	■ 0.0005	■ 0.0002

▶ Tap the toggle to enable or disable **Subscript** in the axis label

▶ Select the encoder for **Input One**

▶ Select the **Couple Operation**

- ■ **Off**
- ■ **+**
- ■ **-**

▶ If **Couple Operation** is enabled, select the encoder for **Input Two**

▶ Tap the toggle to enable or disable **Show axis**

▶ Tap the **Next** button

▶ Repeat the previous steps to setup each display

▶ Tap the **Save & Continue** button

5.3 Machine Tools

Machine Tool settings allow you to:

- Add a machine tool
- Edit an existing machine tool
- Delete an existing machine tool

5.3.1 Add a machine tool

To add a machine tool:

▶ Tap **Machine Tools**

▶ Tap **+**

▶ Select the desired bluetooth enabled machine tool from the **Available bluetooth devices**

▶ Tap the **Scan Again** button if the required device is not listed in the **Available bluetooth devices**

Once connected to the machine tool:

▶ Tap the **Step 3** button to continue and begin the machine tool setup

▶ Select the application
  - **Mill**
  - **Turn**

▶ Enter a **Machine tool name ***

▶ Select the **Number of Encoders**

▶ Select the **Default unit** of measure
  - **inch**
  - **mm**

▶ Tap the **Save & Continue** button

▶ Tap the **Step 4** button to continue and configure the encoders

▶ Select the **Encoder Type**
  - **Linear**
  - **Rotary**
  - **Linear (rotary)**

▶ Select the **Encoder Resolution (µm)** or enter using the keypad

▶ Select the **Reference mark** type
  - **Single**
  - **CODED/2000**
  - **EverTrack**
  - **P-trac**
  - **Position trac**
  - **None**

▶ Select the **Count Direction**
  - **Positive**
  - **Negative**

▶ Tap the toggle to enable or disable counting **Error monitor**

▶ Tap the **Next** button

▶ Repeat the previous steps to configure each encoder

▶ Tap the **Save & Continue** button

▶ Tap the **Step 5** button to continue and configure the encoder displays

| X | | ▶ | Tap the axis button and select an axis label |

- ▪ X
- ▪ Y
- ▪ Z
- ▪ U
- ▪ V
- ▪ W
- ▪ A
- ▪ B
- ▪ C
- ▪ S
- ▪ T
- ▪ Q

▶ Select the **Display Resolution**

- ▪ **0.05**
- ▪ **0.02**
- ▪ **0.01**
- ▪ **0.005**
- ▪ **0.002**
- ▪ **0.001**
- ▪ **0.0005**
- ▪ **0.0002**

▶ Tap the toggle to enable or disable **Subscript** in the axis label

▶ Select the encoder for **Input One**

▶ Select the **Couple Operation**

- ▪ **Off**
- ▪ **+**
- ▪ **-**

▶ If **Couple Operation** is enabled, select the encoder for **Input Two**

▶ Tap the toggle to enable or disable **Show axis**

▶ Tap the **Next** button

▶ Repeat the previous steps to setup each display

▶ Tap the **Save & Continue** button

5.3.2 Edit Machine Tool

To edit a machine tool:

▶ Tap **Machine Tools**

▶ Tap the machine tool you would like to edit

▶ Edit the **Machine name *** if required

Bluetooth Pair

▶ Tap the toggle to enable or disable **Bluetooth Pair** with an IBT400 or IBT400+ product

Position recall

Position recall will store the last position of each axis when power was turned off, and then redisplay that position once power is turned back on.

> ℹ️ Any movement that occurs while power is off will be lost. Whenever power has been off it is recommended to re-establish workpiece datums using the reference mark evaluation procedure.
> **Further information:** "Reference mark evaluation", Page 29

▶ Tap the toggle to enable or disable **Position recall**

Default unit

▶ Tap the toggle to select the **Default unit** of measure

- ▪ **inch**
- ▪ **mm**

Encoders

▶ Tap **Encoders**

▶ Select the **Encoder Type**
- ■ **Linear**
- ■ **Rotary**
- ■ **Linear (rotary)**

▶ Select the **Encoder Resolution (µm)** or enter using the keypad

▶ Select the **Reference mark** type
- ■ **Single**
- ■ **CODED/2000**
- ■ **EverTrack**
- ■ **P-trac**
- ■ **Position trac**
- ■ **None**

▶ Select the **Count Direction**
- ■ **Positive**
- ■ **Negative**

▶ Tap the toggle to enable or disable counting **Error monitor**

▶ Repeat the previous steps to configure each encoder

▶ Tap the **Save** button

or

▶ Tap **Discard** to discard the changes

Displays

▶ Tap **Displays**

▶ Tap the axis button to set the axis label

■ **X**	■ **Y**	■ **Z**	■ **U**
■ **V**	■ **W**	■ **A**	■ **B**
■ **C**	■ **S**	■ **T**	■ **Q**

▶ Select the **Display Resolution**

■ **0.05**	■ **0.02**	■ **0.01**	■ **0.005**
■ **0.002**	■ **0.001**	■ **0.0005**	■ **0.0002**

▶ Tap the toggle to enable or disable **Subscript** in the axis label

▶ Select encoder **Input One**
- ■ **X1 – X6**

▶ Select the **Couple Operation**
- ■ **Off**
- ■ **+**
- ■ **-**

▶ If **Couple Operation** is enabled, select encoder **Input Two**
- ■ **X1 – X6**

▶ Tap the toggle to enable or disable **Show axis**

▶ Repeat the previous steps to setup each display

▶ Tap the **Save** button

or

▶ Tap **Cancel** to cancel the changes

Error Compensation

Error Compensation is used to compensate for encoder and machine travel variations with error correction coefficients. Coefficients are determined by comparing actual measurements of a standard to the nominal values.

For machine tools, the distance a cutting tool travels, measured by an encoder, can in certain cases differ from the actual tool travel. This error can occur due to ball screw pitch error, or deflection and tilting of axes. Errors can be determined with a reference measurement system, such as gauge blocks.

The product provides the ability to compensate for linear errors, and each axis can be programmed separately with the appropriate compensation.

> **ⓘ** Error compensation is only available when using linear encoders.

> **ⓘ** The adjustment procedure is the same for each axis. The following section describes the configuration of one axis. Repeat the procedure for each axis.

Configuring Linear Error Compensation

Linear Error Compensation (LEC) can be applied, if the results of a comparison with a reference standard show a linear deviation over the whole measuring length. In this case the error can be compensated by the calculation of a single correction factor.

To calculate the correction factor use this formula:

Correction factor **LEC = ((S − M) / M) x 10^6 ppm** with:

S = measured length with a reference standard

M = measured length with device at axis

Example:

If the length of the standard you used is 500 mm, and the measured length along the X-axis is 499.95, then the LEC for the X-axis is 100 parts per million (ppm).

LEC = ((500 − 499.95) / 499.95) x 10^6 ppm = 100 ppm (rounded to the nearest whole number).

To configure LEC:

▶ Tap **Error Compensation**

▶ Select the encoder to configure

▶ Select **Linear** from the **Type of compensation** pull-down

▶ Enter the **Measured** length using the keypad

  or

🎓   ▶ Tap the Teach button

▶ Enter the **Actual** length using the keypad

▶ Repeat these steps for each axis you want to configure

▶ Tap the **Save** button to save the changes

> droPWR calculates the LEC factor

  or

▶ Tap **Discard** to discard the changes

Backlash Compensation

When using a rotary encoder with a lead screw, a change in direction of the table might cause an error in the displayed position due to clearances within the lead screw assembly. This clearance is referred to as backlash. This error can be compensated for by entering the amount of backlash using the **Backlash Compensation** feature.

If the rotary encoder is ahead of the table (displayed value is greater than the table's true position), this is called positive backlash, and the value entered should be the positive value of the amount of error.

No Backlash Compensation is 0.000.

To specify backlash compensation:

▶ Tap **Backlash Compensation**

▶ Select the encoder to configure

▶ Select **On** from the **Type of compensation** pull-down

▶ Enter the positive value for the amount of error in the **Length** field

▶ Repeat these steps for each axis you want to configure

▶ Tap the **Save** button to save the changes

  or

▶ Tap **Discard** to discard the changes

5.3.3 Remove a machine tool

To remove a machine tool:

▶ Tap the Edit icon

▶ Tap **-** next to the machine tool you want to remove

▶ Tap the **Delete** button to confirm deleting the machine tool

5.4 Settings

5.4.1 My Profile

My Profile settings allow you to edit the user settings for your profile.

My Profile settings include:

- **Personal**
- **Set Language**

  To edit My Profile settings:

▶ Tap **Settings**

▶ Tap **My Profile**

Personal

Personal settings allow you to edit the following:

- ■ **Screen name**
- ■ **PIN code**
- ■ **Email Address**
- ■ **Avatar color**

  To edit your personal settings:

- ▶ Tap **Personal**
- ▶ Tap the **Screen name** field and use the keypad to edit your screen name
- ▶ Tap the **PIN code** field and use the keypad to edit your code
- ▶ Tap the **Email Address** field and use the keypad to edit your Email address
- ▶ Tap the **Require PIN** toggle to enable or disable requiring a PIN code to open the app
- ▶ Tap an **Avatar color** to select the color you would like for your avatar

Settings

Settings is used to open the iPad settings application and edit settings that effect the droPWR application.

To open the iPad settings:

▶ Tap the icon next to **Settings**

- ▶ Tap **Open settings** to confirm that you want to open the iPad settings
- \> The iPad settings application opens.

Changing the language

The default language is English.

> 🛈 droPWR is available in the following languages:
> - ■ **English**
> - ■ **French**
> - ■ **German**
> - ■ **Spanish**
>
> **English** will be displayed if a language other than one listed is selected.

To change the Language:

- ▶ Tap **General**
- ▶ Tap **Language & Region**
- ▶ Tap **iPad Language**
- ▶ Select the required language

> 📖 For additional information on setting the language, refer to the iPad User Guide for your iPadOS version.

Changing the appearance

Display & Brightness is used to set the appearance of the user interface. Select an appearance that makes the interface easiest to view in your workspace lighting conditions.

To select an appearance:

▶ Tap **Display & Brightness**

▶ Tap **Light** or **Dark** to select an appearance

or

▶ Tap the **Automatic** toggle to enable or disable automatic appearance based on ambient light levels

> 📖 For additional information on setting the display appearance, refer to the iPad User Guide for your iPadOS version.

5.4.2 App Settings

App Settings allow you to import or export data files, and provides access to www.acu-ritesolutions.com.

Data file types:

- **Machine Settings**
- **Tool Tables**

Accessing App Settings

To access App Settings:

▶ Tap **Settings**

▶ Tap **App Settings**

Import

Machine settings can be imported from an exported data file or from an existing Machine Tool. Other data can be imported from an exported data file. **Further information:** "Export", Page 44

Importing settings from an exported file:

To import a settings file:

▶ Open droPWR

▶ Connect to an IBT unit

▶ Press the **Home** button to move droPWR to the background

▶ Open the **Files** app and browse to the location of the desired settings file

▶ Open the settings file

▶ Tap the Share icon

▶ Tap the droPWR icon

▶ Tap **Import**

▶ Tap **Done** in the file viewer

> The currently connected Machine Tool will import the data from the file

Importing settings from an existing Machine Tool

To import settings from an existing Machine Tool:

▶ Open droPWR

▶ Tap **Machine Tools**

▶ Tap **+**

▶ Connect to a new IBT unit

▶ Tap **Import**

▶ Select an existing Machine Tool to import its settings

▶ Tap **Import**

> The imported settings are applied to the new Machine Tool

Export

Exporting a settings file

Two types of files can be exported:

■ **Export Machine Settings**: Exports Machine Tool's settings

■ **Export Tool Table**: Exports the active Machine Tool's tool table

To export a settings file:

▶ Tap **Export**

▶ Tap the file type you would like to export

   ■ **Export Machine Settings**

   ■ **Export Tool Table**

> A confirmation notice will appear

▶ Tap **Export**

▶ Tap **Save to Files**

▶ Create a new folder

   or

▶ Tap the folder where you want to store the file

▶ Tap **Save**

> A confirmation notice will appear

▶ Tap **OK**

Visit www.acu-ritesolutions.com

Visit www.acu-ritesolutions.com provides access to the Acu-Rite Solutions website where you can download the droPWR Operating Instructions, register products, and more.

To open www.acu-ritesolutions.com:

▶ Tap **Visit www.acu-ritesolutions.com**

▶ Tap **Visit**

> www.acu-ritesolutions.com opens in the default browser

5.5 Subscription

5.5.1 Subscription

The following subscription options are available during initial setup in the setup guide:

- **IBT Devices Starter** This subcription covers the first year of use
- **Demo mode (free)** The demonstration mode allows you to access and sample the features of droPWR without being connected to an IBT device

The following subscription options are available after the initial setup using the Setup guide:

- **IBT Devices Starter** This subcription for the first year of use is still available if Demo mode was selected in the Setup guide
- **1-year subscription** This subcription covers 1 year of use and needs to be renewed each year
- **IBT Devices Lifetime Use** This subcription provides lifetime use of the app

To select a subscription:

▶ Tap **Subscription**

▶ Tap **Subscription**

▶ Tap the subscription you would like to select
▶ Tap **Subscribe** in the verification window and follow the on screen instructions

5.6 View Settings

View Settings allow you to adjust the display configuration and to enable or disable Zoom active axes and Highlight active axes.

To access View Settings:

▶ Tap the **View Settings** icon in the upper right corner of the Readout view

Hiding an axis

To hide an axis on the display:

▶ Tap the label of an axis under **Axes displayed in view**
A visible axis is white, a hidden axis is black.

Zoom Timeout

The amount of time a zoomed axis stays zoomed when **Zoom active axes** is enabled.

To set the zoom timeout:

▶ Enter the number of seconds the active axes stays zoomed
- 1 - 10 seconds

Zoom active axes

The axis in motion is displayed larger than axes not in motion

To enable or disable zooming the active axes:

▶ Tap the toggle for **Zoom active axes**

Highlight active axes

In Day mode axes in motion are displayed in black, axes not in motion are displayed in gray. In Night mode axes in motion are displayed in white, axes not in motion are displayed in gray.

To enable or disable highlighting the active axes:

▶ Tap the toggle to enable or disable **Highlight active axes**

Saving View Settings

▶ Tap the **Save** button to save the changes

or

▶ Tap **Discard** to discard the changes

6

Milling specific operations

6.1 Overview

This chapter describes operations and features specific to milling applications.

> **i** Make sure that you have read and understood the "Basic Operation" chapter before performing the activities described in this section.
> **Further information:** "Basic operation", Page 23

6.2 1/2 button

The 1/2 button is available when the product is setup for milling applications and is used to find the centerline (or midpoint) between two locations along a selected axis of a workpiece. This can be performed in either Actual Value or Distance-To-Go mode.

> **i** This feature will change datum locations when in Actual Value mode.

Example: Finding the midpoint along a selected axis
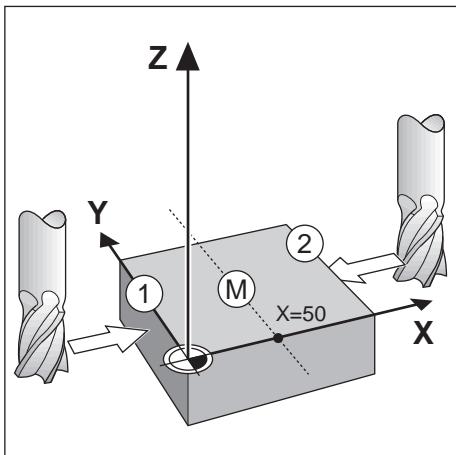
X dimension: X = 100 mm

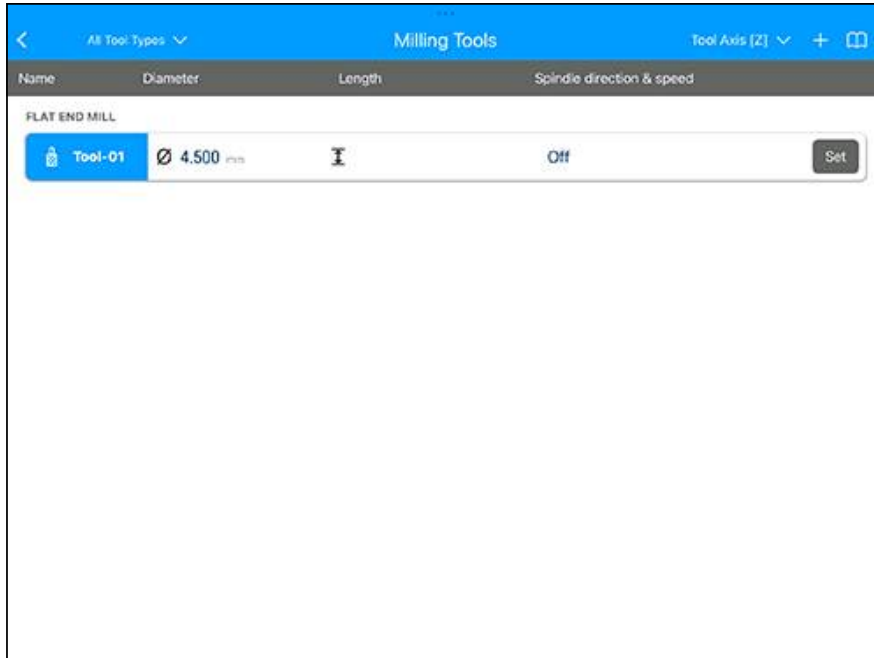Midpoint: 50 mm

To find the midpoint:

▶ Move the tool to the first point
▶ Tap the **X** axis button
▶ Tap the **zero** button
▶ Move to the second point
▶ Tap the **1/2** button
▶ Move the tool until zero is reached
> This is the midpoint location

6.3 Tool Table

The Tool Table is used to store diameter and length offset information for tools. The table can store information for up to 99 tools.

Opening the tool table

To open the tool table:

▶ Tap the **Set Tool** button

> The Tool Table opens.

6.3.1 Tool compensation

Tool compensation allows you to enter workpiece dimensions directly from the drawing.

Radius compensation

Tool radius compensation is calculated based on the value entered in the Diameter field in the Tool form.

R represents the tool radius. The displayed distance-to-go is automatically lengthened **R+**, or shortened **R-** by the value of the tool radius.

Further information: "Presetting a target position", Page 61
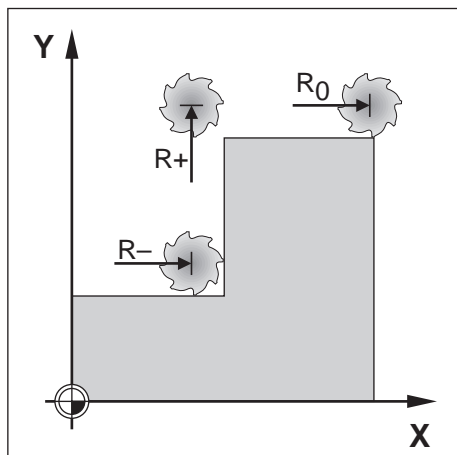
Length offset

Tool length offset is calculated based on the value entered into the Length field in the Tool form. The length offset can be entered as a known value or the product can teach the offset.

Further information: "Entering tool data", Page 51

The tool length is the difference in length ΔL between the tool and the reference tool. The length difference is indicated with the "Δ" symbol. The reference tool is indicated by T1.

- If the tool is **longer** than the reference tool: ΔL > 0 (+)
- If the tool is **shorter** than the reference tool: ΔL < 0 (−)
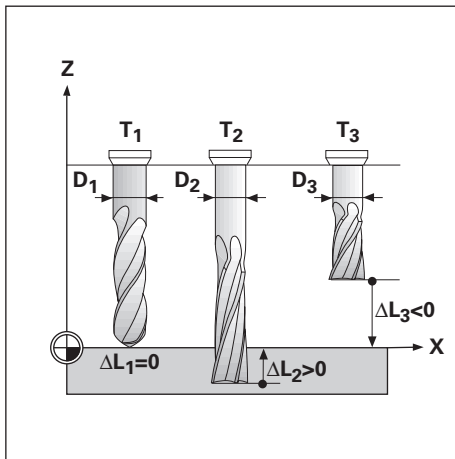
6.3.2 Entering tool data

To enter a tool:

▶ Tap the **Set Tool** button

▶ Tap **+**
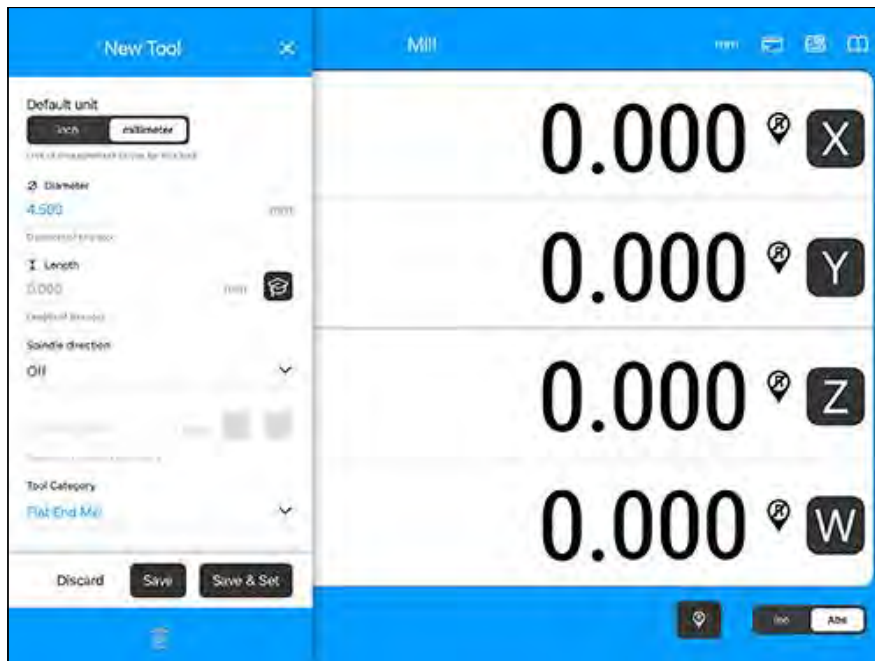
▶ Enter a tool number between 01 - 99

▶ Select the **Default unit** of measure

   ▪ **inch**

   ▪ **mm**

▶ Enter the tool **Diameter**

▶ Enter the tool **Length**

▶ or

[icon]   ▶ Tap the Teach button and follow the procedure for Teaching length offset described in this section

Spindle data assists the operator with direction and speed information.

▶ Select the **Spindle direction**

   ▪ **Off**

   ▪ **Reverse**

   ▪ **Forward**

▶ Enter the **Spindle Speed**

or

[icon]   ▶ Tap the RPM icon to calculate the spindle speed

or

[icon]   ▶ Tap the Teach icon to teach the spindle speed

▶ Select the **Tool Category**

▪ **Not defined**	▪ **Counter Sink**	▪ **Reamer**
▪ **Ball End Mill**	▪ **Drill**	▪ **Roughing Mill**
▪ **Boring Head**	▪ **Engraving Tip**	▪ **Shell End Mill**
▪ **Broach**	▪ **Flat End Mill**	▪ **Special Mill**
▪ **Carbide Mill**	▪ **Fly Cutter**	▪ **Tap**
▪ **Counter Bore**	▪ **Pilot Drill**	

Teaching length offset

It is also possible to have the product determine an offset. This method involves touching the tip of each tool to a common reference surface. This allows the product to determine the difference between the length of each tool.

> **ℹ** Only the tools set using the same reference surface may be changed without having to reset the datum.

> **ℹ** If the tool table already contains tools in which the length has been set, the reference surface should first be established using one of them. If not, you will not be able to switch between the new tools and the existing tools without having to re-establish the datum. Before adding the new tools, select one of the tools from the tool table. Touch the tool to a reference surface and set the datum to 0

To teach Length Offset:

▶ Move the tool until its tip is touching the reference surface

> **🎓**
> ▶ Tap the Teach button
> > The product will calculate an offset relative to the reference surface

▶ Repeat the procedure for each additional tool using the same reference surface

6.3.3 Setting a tool

Before you start machining, set the tool you are using from the Tool Table. The product then takes into account the stored tool data when working with tool compensation.

To set a tool:

▶ Tap the **Set Tool** button

▶ Tap the **Set** button next to the tool you would like to use

6.3.4 Deleting a tool

Tools that are no longer needed can be deleted from the Tool Table.

To delete a tool:

▶ Tap the **Set Tool** button

▶ Tap the tool you would like to delete

> **🗑**
> ▶ Tap the **Trash** icon

▶ Tap **Delete** to confirm deleting the tool

6.3.5 Importing and exporting a tool table

A Tool Table can be imported from an existing file or exported for backup and future use.

Further information: "App Settings", Page 43

6.4 Datums

Datum settings define the relationships between the axis positions and the display values.

The easiest way to create datum points is to use the probing function when probing the workpiece with the edge of a tool. Datum points can also be created by touching the edges of the workpiece, one after the other with a tool, and manually entering the tool positions as datum points.

The datum table can hold up to 99 datum points. This will eliminate from having to calculate the axis travel when working with complicated workpiece drawings containing several datums.

6.4.1 Creating a workpiece datum without using the probing function
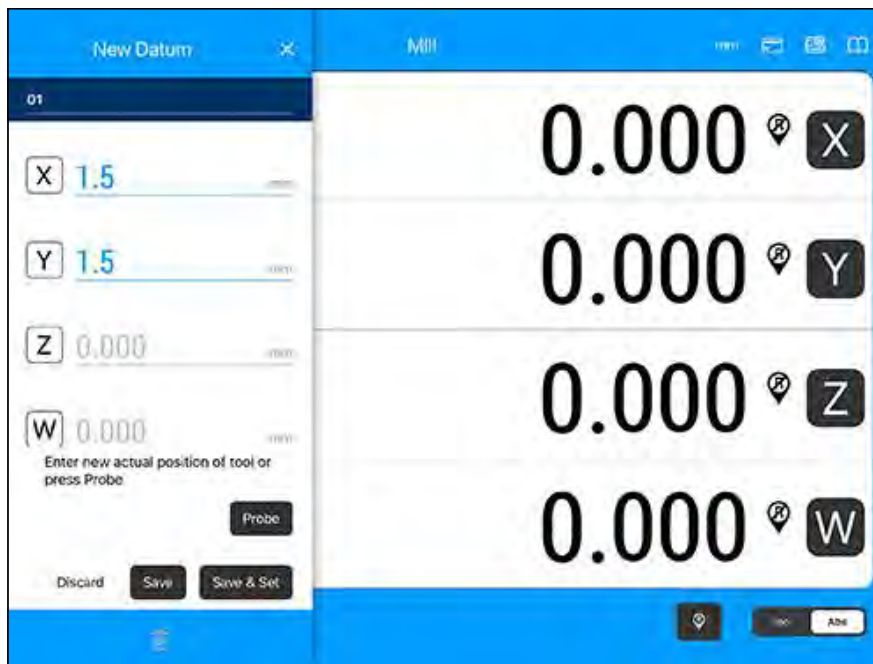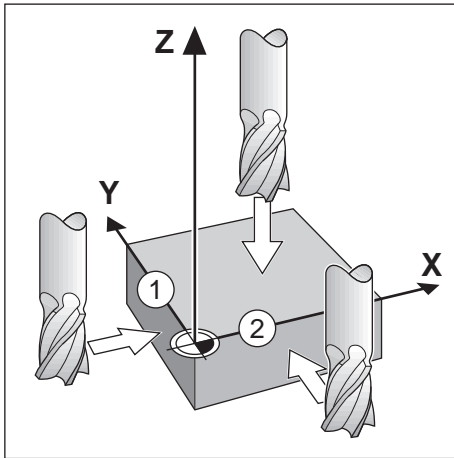
- Axis sequence in this example: X - Y - Z

To create a datum without the probing function:

▶ Tap the **Set datum** button

▶ Enter the a datum number

▶ Highlight the **X** axis field

▶ Touch the workpiece at edge **1**

▶ Enter the position of the tool center (X = 1.5 mm)

▶ Highlight the **Y** axis field

▶ Touch the workpiece at edge **2**

▶ Enter the position of the tool center (Y = 1.5 mm)

▶ Highlight the **Z** axis field

▶ Touch the workpiece surface

▶ Enter the position of the tool tip (Z = 0 mm) for the Z coordinate of the datum

▶ Tap the **Save** button to save the datum

  or

▶ Tap the **Save & Set** button to save the datum and set it as the current datum
or
▶ Tap **Discard** to discard the changes

6.4.2 Probing with a tool

A tool or non-electrical edge finder can be used to create datum points.
The following probing functions are available:
- Workpiece edge as datum: **Edge** button
- Centerline between two workpiece edges: **Center Line** button
- Center of a hole or cylinder: **Circle Center** button

In all probing functions the product takes into account the current tool's tip diameter.

Probing an Edge with a tool

To probe an edge with a tool:

▶ Set the active tool to the tool that will be used to set the datum

▶ Tap the **Set datum** button

▶ Enter the datum number

▶ Highlight the **X** axis field

▶ Tap the **Probe** button

▶ Tap the **Edge** button

▶ Touch the workpiece edge

▶ Tap the Teach button

The Teach button is useful when determining tool data by touching the workpiece in the absence of an edge finder with feedback. To avoid losing the position value when the tool is retracted, tap the Teach button to store the value while it is in contact with the workpiece edge. The location for the touched edge will take into account the diameter of the tool in use and the last direction the tool was moved prior to tapping the Teach button.

▶ Retract the tool from the workpiece

▶ Enter the position of the workpiece edge

▶ Tap the **Save** button to save the datum

or

▶ Tap the **Save & Set** button to save the datum and set it as the current datum

or

▶ Tap **Discard** to discard the changes

Probing a Center Line with a tool
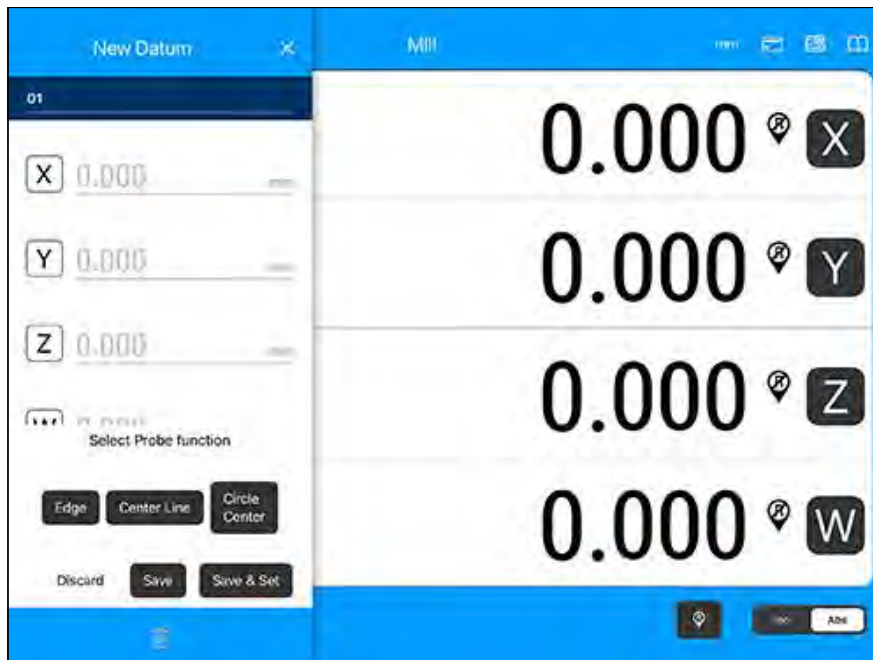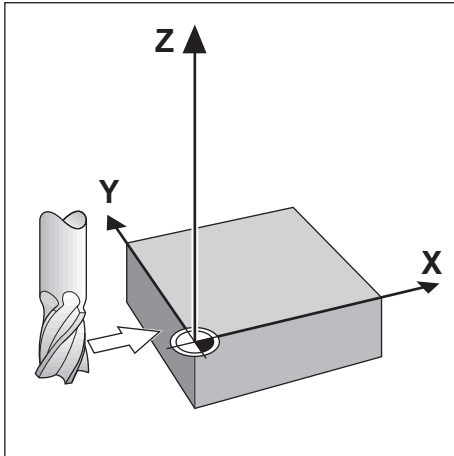

To probe a centerline with a tool:

▶ Set the active tool to the tool that will be used to set the datum
▶ Tap the **Set datum** button
▶ Enter the datum number
▶ Highlight the **X** axis field
▶ Tap the **Probe** button
▶ Tap the **Center Line** button
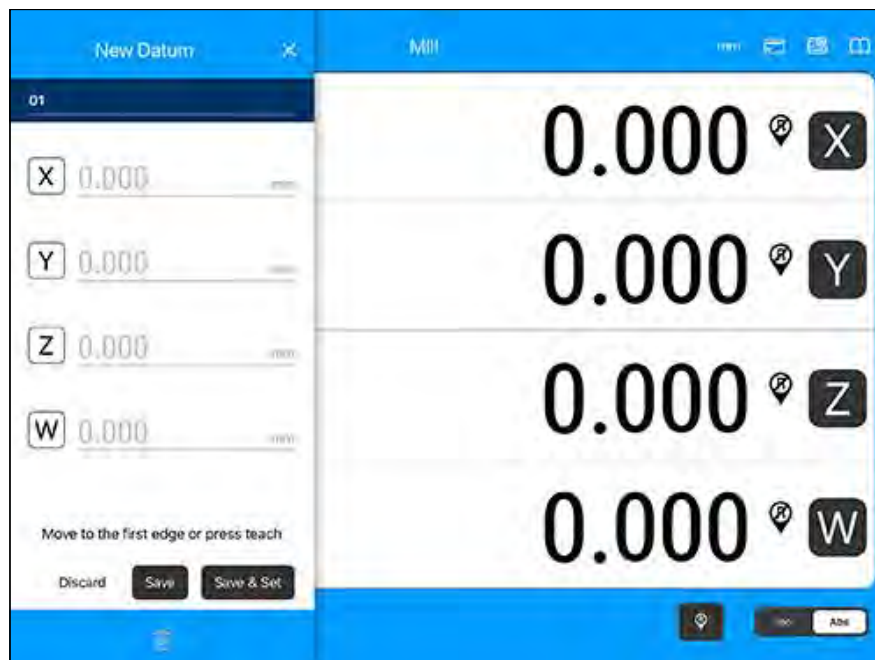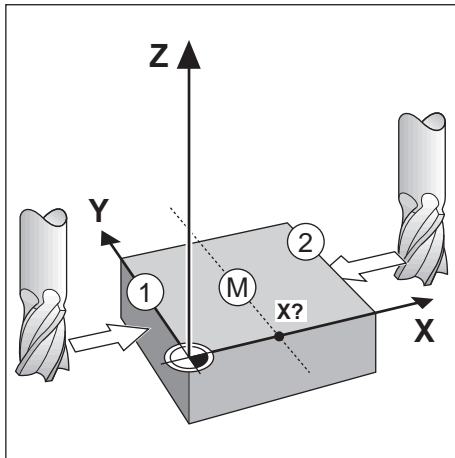▶ Touch the first workpiece edge **1**

> ▶ Tap the Teach button

▶ Touch the second workpiece edge **2**

> ▶ Tap the Teach button

> The datum is set to 0.000 and the distance between the edges is displayed
▶ Retract the tool from the workpiece
▶ Enter the position of the workpiece centerline

▶ Tap the **Save** button to save the datum
   or
▶ Tap the **Save & Set** button to save the datum and set it as the current datum
   or
▶ Tap **Discard** to discard the changes

Probing a Circle Center with a tool
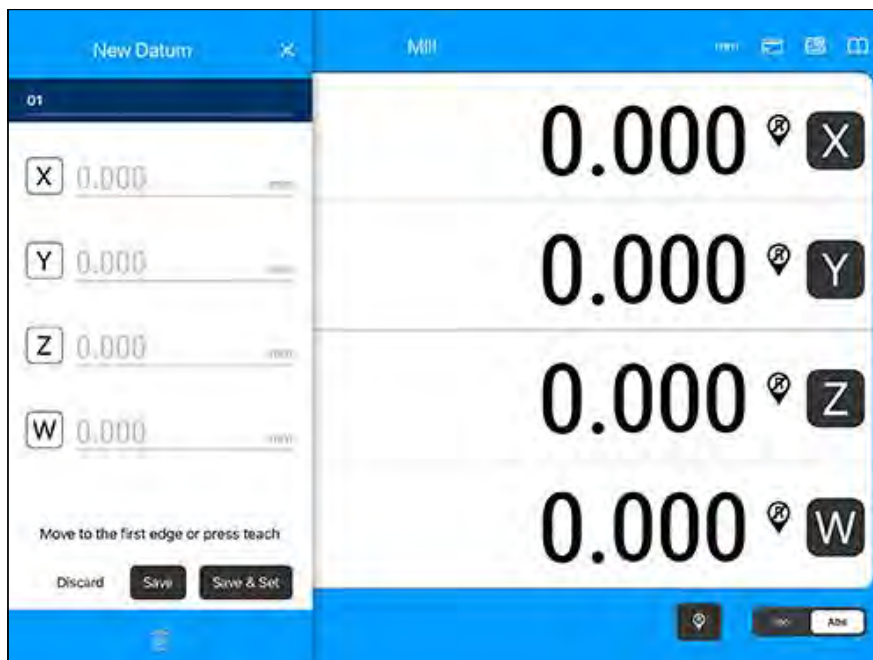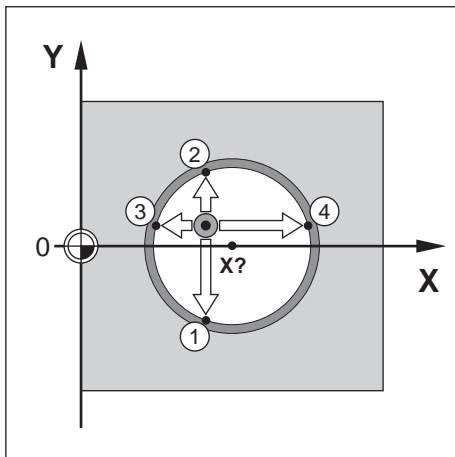
To probe a Circle Center with a tool:

▶ Set the active tool to the tool that will be used to set the datum

▶ Tap the **Set datum** button

▶ Enter the datum number

▶ Highlight the **X** axis field

▶ Tap the **Probe** button

▶ Tap the **Circle Center** button

▶ Touch the first workpiece edge **1**

           ▶ Tap the Teach button

▶ Touch the second workpiece edge **2**

           ▶ Tap the Teach button

▶ Touch the third workpiece edge **3**

▶ Tap the Teach button

▶ Touch the fourth workpiece edge **4**

▶ Tap the Teach button

> The **X** and **Y** datums are set to 0.000 and the diameter of the circle is displayed.

▶ Retract the tool from the workpiece

▶ Enter the **X** and **Y** axis position of the circle center

▶ Tap the **Save** button to save the datum

or

▶ Tap the **Save & Set** button to save the datum and set it as the current datum

or

▶ Tap **Discard** to discard the changes

6.5 Presetting a target position

The Preset function allows you to indicate the nominal (target) position for the next move. Once the new nominal position information is set the display will switch to Distance-To-Go mode and show the distance between the current position and the nominal position. You now only need to move the table until the display is zero and you will be at the required nominal position. The information for the location of the nominal position can be entered as an absolute move from the current datum zero or as an incremental move from the current nominal position.

Presetting also allows you to indicate which side of the tool will be doing the machining at the nominal position. The R0, R+, R-, and I buttons in the Preset form define the offset that will be in effect during the move. **R+** indicates that the center line of the current tool is in a more positive direction than the edge of the tool. **R-** indicates that the center line is in a more negative direction than the edge of the current tool. Using offsets automatically adjusts the distance-to-go value to account for the diameter of the tool. **I** indicates an incremental move.

The Prest table can hold up to 99 presets.
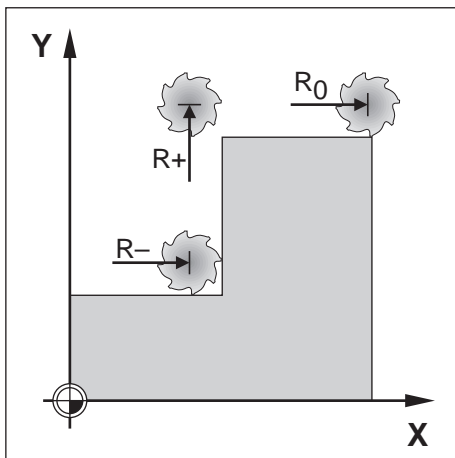
6.5.1 Absolute distance preset

Example

Milling a shoulder by traversing to display value zero using absolute position.
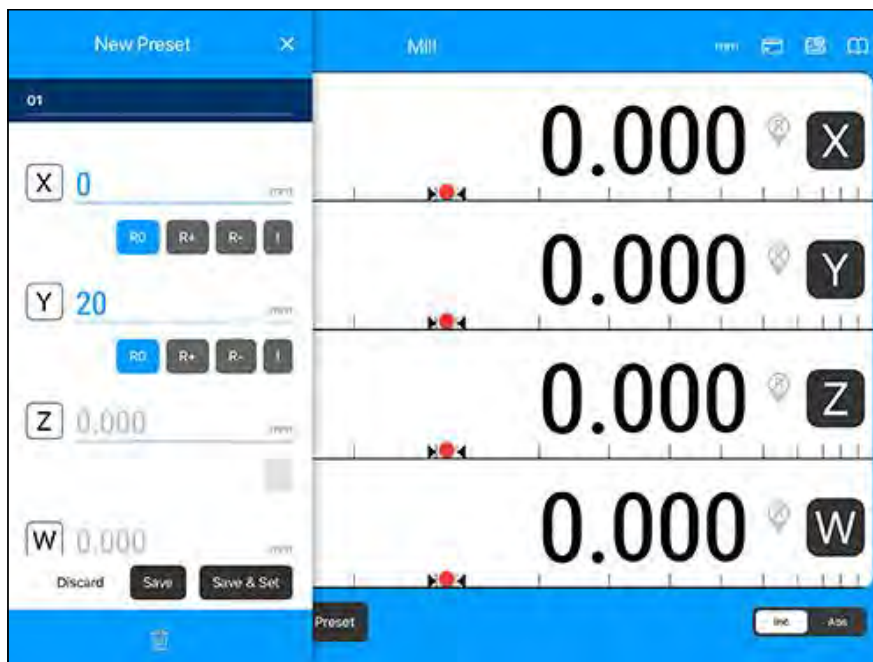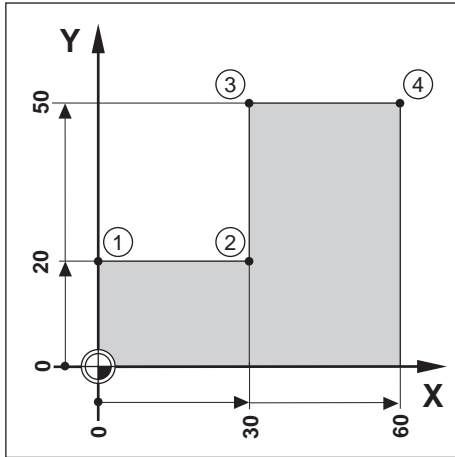
The coordinates are entered as absolute dimensions; the datum is the workpiece zero. Using this figure example:

- Corner **1**: X = 0 / Y = 20
- Corner **2**: X = 30 / Y = 20
- Corner **3**: X = 30 / Y = 50
- Corner **4**: X = 60 / Y = 50

▶ Select the tool with the appropriate tool data

▶ Pre-position the tool to an appropriate location (such as X = Y = -10)

▶ Move the tool to milling depth

▶ Tap the **Preset** button

▶ Tap **+** to add a preset

▶ Enter a preset number between 01 - 99

▶ Enter the nominal position value for corner point **1**: Y = 20

- ▶ Tap **R +**
- ▶ Tap the **Save & Set** button
- ▶ Traverse the **Y** axis until the display value is zero
- > The graphic positioning aid is now centered between the two center marks.
- ▶ Tap the **X** axis button
- > The X axis menu opens
- ▶ Tap the **Preset** button
- ▶ Enter the nominal position value for corner point **2**: X = 30
- ▶ Tap **R -**
- ▶ Tap the **Save & Set** button
- ▶ Traverse the **X** axis until the display value is zero
- > The graphic positioning aid is now centered between the two center marks.
  Presets can be entered in the same manner for corners **3** and **4**.

6.5.2 Incremental distance preset

Example

Drilling by traversing to display value zero with incremental positioning.

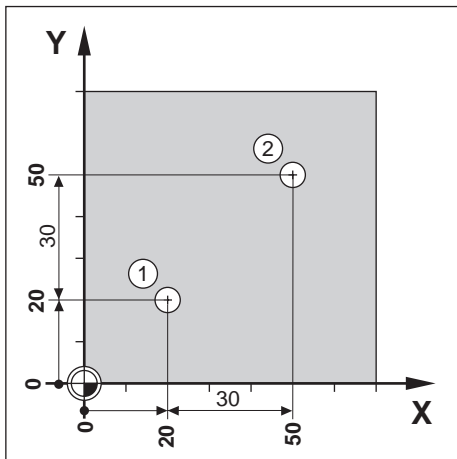> **i** Enter the coordinates in incremental dimensions. These are indicated in the following (and on the screen) with a preceding **I** (Incremental). The datum is the workpiece zero.

- Hole **1** at: X = 20 / Y = 20
- Distance from hole **1** to hole **2**: XI = 30 / YI = 30
- Hole depth: Z = −20
- Operating mode: Distance-To-Go (Inc)

To preset the location for hole **1**:

▶ Tap the **Preset** button

▶ Tap **+** to add a preset

▶ Enter a preset number between 01 - 99

▶ Enter the nominal position value for hole **1**: X = 20

▶ Tap **R0**

Note that these presets are Absolute Presets.

▶ Enter the nominal position value for hole **1**: Y = 20

▶ Tap **R0**

▶ Enter the nominal position value for the hole depth: Z = -20

▶ Tap the **Save & Set** button

▶ Drill hole **1**: Traverse the **X**, **Y**, and **Z** axis until the display value is zero

> The graphic positioning aids are now centered between the two center marks

▶ Retract the drill

To preset the location for hole **2**:

▶ Tap the **X** axis button

> The X axis menu opens

▶ Tap the **Preset** button

▶ Enter the nominal position value for hole **2**: X = 30

▶ Tap **I** to mark your input as an incremental dimension

▶ Enter the nominal position value for hole **2**: Y = 30

▶ Tap **I** to mark your input as an incremental dimension

▶ Tap the **Save & Set** button

▶ Drill hole **2**: Traverse the **X**, **Y**, and **Z** axis until the display value is zero

> The graphic positioning aids are now centered between the two center marks

▶ Retract the drill

6.6 Features

Tapping the **Feature** button provides access to the **Circle Patterns**, **Linear Patterns**, **Incline Mill**, and **Arc Mill** milling features.

The **Circle Patterns** and **Linear Patterns** features provide ways to calculate and machine various hole patterns. The **Incline Mill** and **Arc Mill** features provide ways to machine a flat diagonal surface, or a rounded surface using a manual machine.

6.6.1 Circle and linear patterns

This section describes circle and linear pattern tables and capabilities. The product provides storage of 99 user definable patterns each for circle and linear. Once patterns are defined, they can be recalled, and executed from the DRO, or from a program.

To access the **Circle Patterns** or **Linear Patterns** table:

▶ Tap the **Feature** button

> The **Circle Patterns** and **Linear Patterns** tabs are shown

▶ Tap the **Circle Patterns** tab to access the **Circle Patterns** table

or

▶ Tap the **Linear Patterns** tab to access the **Linear Patterns** table

> The corresponding hole pattern table will open

Circle and linear hole patterns

Required Circle Patterns information
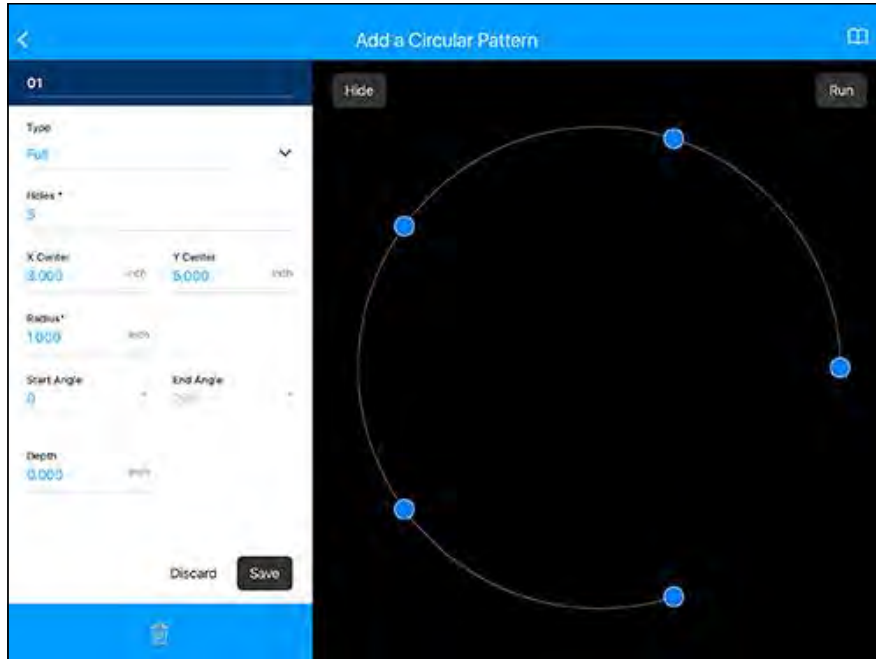
- ■ **Type**: type of pattern, **Full** or **Segmented**
- ■ **Holes***: number of holes in the pattern
- ■ **X Center**: X axis location of the center of the hole pattern
- ■ **Y Center**: Y axis location of the center of the hole pattern
- ■ **Radius***: radius of the pattern
- ■ **Start Angle**: angle between X axis and the first hole
- ■ **End Angle**: angle between the X axis and the last hole
- ■ **Depth**: the target depth for drilling in the tool axis

Required Linear Patterns information
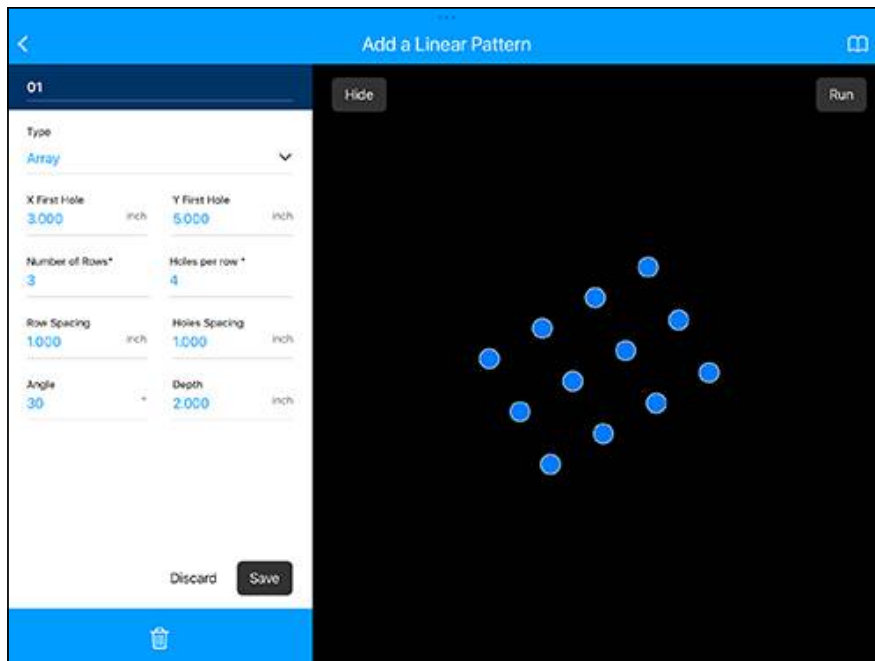
- ■ **Type**: type of pattern, **Array** or **Frame**
- ■ **X First Hole**: X axis location of the first hole of the pattern
- ■ **Y First Hole**: Y axis location of the first hole of the pattern
- ■ **Number of Rows***: the number of rows in the pattern
- ■ **Holes per row***: number of holes in each row of a pattern
- ■ **Row Spacing**: the spacing between each row of the pattern
- ■ **Hole Spacing**: the spacing between each hole in a row
- ■ **Angle**: the angle or rotation of the pattern
- ■ **Depth**: the target depth for drilling in the tool axis

Creating a pattern

To create a pattern:

▶ Tap the **Feature** button
▶ Tap the **Circle Patterns** or **Linear Patterns** tab
> The tab will display any patterns that have been previously defined
▶ Tap **+** to create a new pattern
> The pattern form is opened
▶ Enter the information to define the pattern
▶ Tap the **Save** button to save the pattern
> The pattern will be saved in the tab. It can now be modified, executed, or referred to from a program.
   or
▶ Tap **Discard** to discard the pattern

Editing a pattern

To edit a pattern:

▶ Tap the **Feature** button

▶ Tap the **Circle Patterns** or **Linear Patterns** tab

> The tab will display any patterns that have been previously defined

▶ Tap the pattern that you would like to edit

> The pattern form is opened

▶ Enter the information that you want to edit

▶ Tap the **Save** button to save the pattern

> The pattern changes will be saved.

or

▶ Tap **Discard** to discard the changes

Deleting a pattern

To delete a pattern from the table:

▶ Tap the **Feature** button

▶ Tap the **Circle Patterns** or **Linear Patterns** tab

▶ Tap the the pattern you would like to delete

     ▶ Tap the Trash button

▶ Tap **Delete** to confirm deleting the pattern

Running a pattern
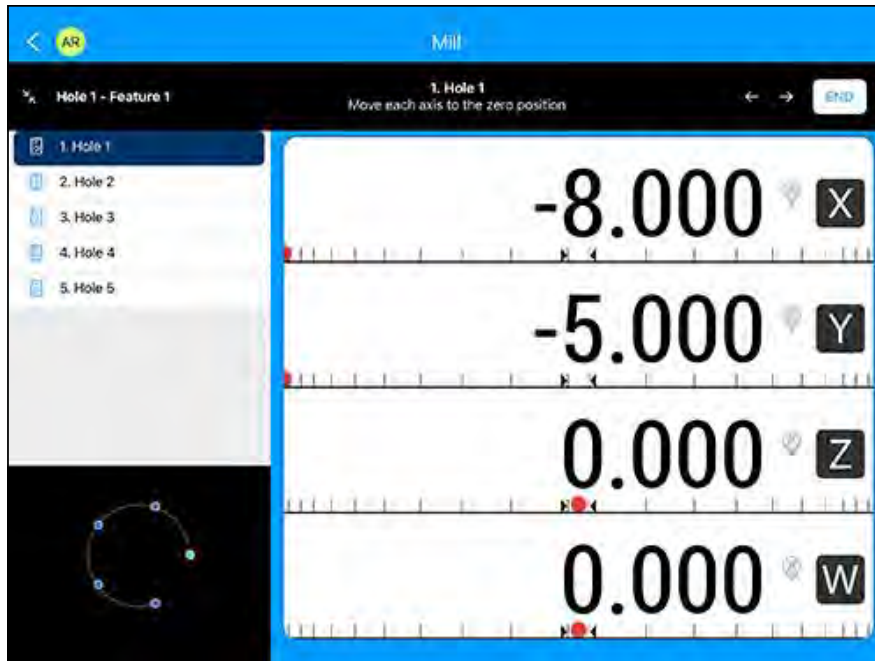
To run a pattern:

▶ Tap the **Feature** button

▶ Tap the **Circle Patterns** or **Linear Patterns** button

▶ Tap the pattern you would like to run

▶ Tap the **Run** button

> The product calculates the positions of the holes and provides a graphical view of the hole pattern

The following buttons are available while running a pattern:

Button	Function
←	Tap ← to select the previous hole in the pattern
→	Tap → to select the next hole in the pattern
End	Tap the **End** button to end execution of the pattern

Example: Entering data and running a circle pattern
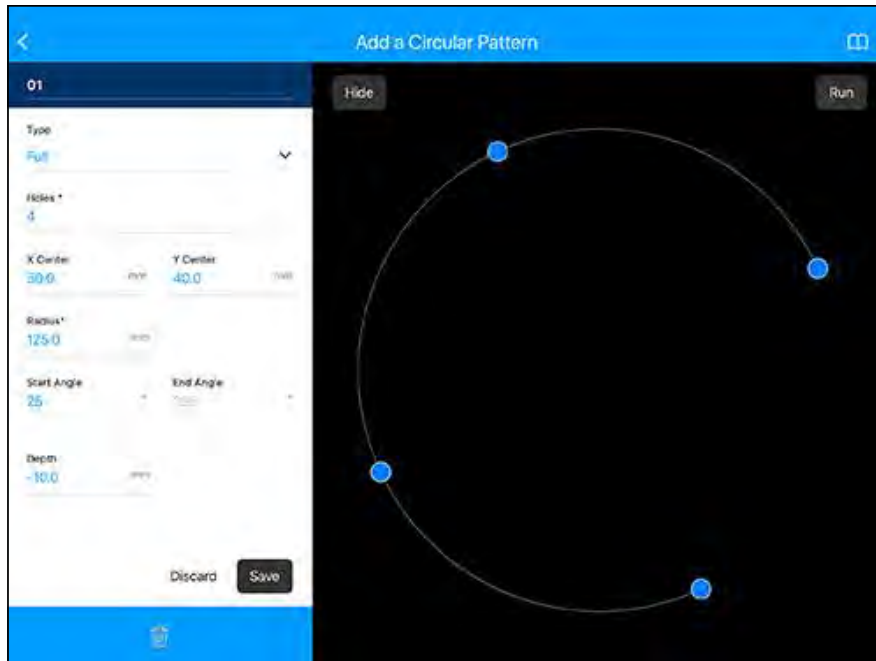
Entering the data:

▶ Tap the **Feature** button

▶ Tap the **Circle Patterns** tab

▶ Tap **+** to create a new pattern

▶ Enter **1** for the pattern number

▶ Select **Full** in the **Type** field

▶ Enter 4 for the number of **Holes***

▶ Enter 50 mm for the **X Center** location

▶ Enter 40 mm for the **Y Center** location

▶ Enter 125 mm for the **Radius*** of the circle pattern

▶ Enter 25° for the **Start Angle**

▶ The **End Angle** is 295° and cannot be changed because the **Type** is **Full**

▶ Enter a **Depth** of -10 mm

   The depth of the hole is optional and may be left blank

▶ Tap the **Save** button to save the pattern

> The **Circle Patterns** tab now shows the pattern that was just defined as pattern
   1
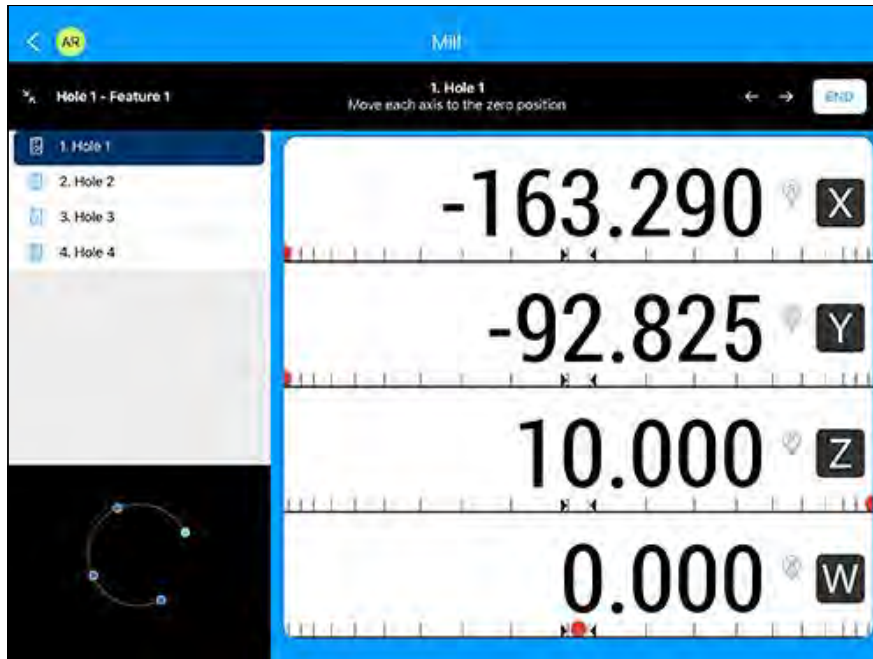
Running the pattern:

▶ Tap the **Run** button

> The Distance-To-Go view is displayed

▶ Move to the hole, move the **X** and **Y** axis until their display values show 0.0

▶ Drill (depth): If a depth was entered into the pattern, move **Z** until it's display value shows 0.0. otherwise, drill to the desired depth

▶ Tap →

▶ Continue to drill the remaining holes in the same manner

When the pattern is complete:

▶ Tap the **End** button

6.6.2   Arc and incline milling

The arc and incline milling features provide ways to machine a rounded surface (arc milling) or a a flat diagonal surface (incline milling) using a manual machine. The product provides storage of 99 user definable features each for arc and incline. Once the features are defined, they can be recalled, and executed from the DRO, or from a program.

To access the **Arc Mill** or **Incline Mill** tab:

▶ Tap the **Feature** button

> The **Arc Mill** and **Incline Mill** tabs are shown

▶ Tap the **Arc Mill** tab to access the **Arc Mill** table

or

▶ Tap the **Incline Mill** tab to access the **Incline Mill** table

> The corresponding mill table will open

Arc and incline milling features

Required Arc Mill information

- **Plane** : the plane to be milled
- **X Center**: X axis center point
- **Y Center**: Y axis center point
- **X Start**: X axis start point
- **Y Start**: Y axis start point
- **X   End**: X axis end point
- **Y   End**: Y axis end point
- **Step**: The distance along the circumference of the arc between each pass, or **Step** along the arc's contour

> The **Step** size is optional. If the value is zero, the operator decides at run-time how far to move between each Step.

Required Incline Mill information

- ■ **Plane**: the plane to be milled
- ■ **X Start**: X axis start point
- ■ **Y Start**: Y axis start point
- ■ **X End**: X axis end point
- ■ **Y End**: Y axis end point
- ■ **Step**: The distance between each pass or each step along the line

> ℹ The **Step** size is optional. If the value is zero, the operator decides at run-time how far to move between each **Step**.

Creating a milling feature

To create a milling feature:

▶ Tap the **Feature** button

▶ Tap the **Arc Mill** or **Incline Mill** tab

> The tab will display any milling features that have been previously defined

▶ Tap **+** to create a new milling feature

> The milling feature form is opened

▶ Enter the information to define the milling feature

▶ Tap the **Save** button to save the milling feature

> The milling feature will be saved in the tab. It can now be modified, executed, or referred to from a program.

or

▶ Tap **Discard** to discard the milling feature

Editing a milling feature

To edit a milling feature:

▶ Tap the **Feature** button

▶ Tap the **Arc Mill** or **Incline Mill** tab

> The tab will display any milling features that have been previously defined

▶ Tap the milling feature that you would like to edit

> The milling feature form is opened

▶ Enter the information that you want to edit

▶ Tap the **Save** button to save the pattern

> The pattern changes will be saved.
   or

▶ Tap **Discard** to discard the changes

Deleting a milling feature

To delete a milling feature:

▶ Tap the **Feature** button

▶ Tap the **Incline Mill** or **Arc Mill** tab

▶ Tap the the milling feature you would like to delete

   ▶ Tap the Trash button

▶ Tap **Delete** to confirm deleting the milling feature

Running a milling feature
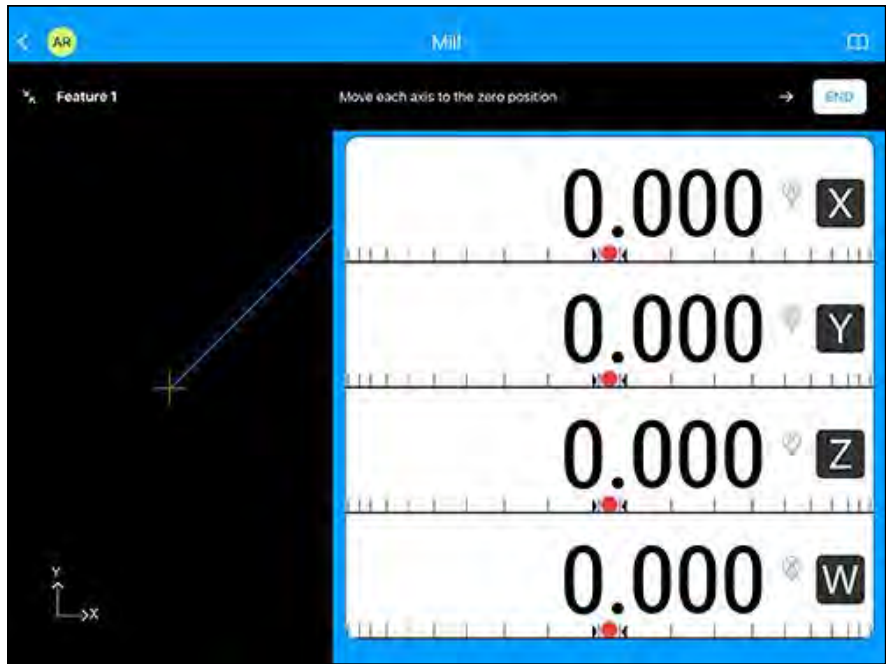
To run a milling feature:

▶ Tap the **Feature** button

▶ Tap the **Incline Mill** or **Arc Mill** tab

▶ Tap the the milling feature you would like to run

▶ Tap the **Run** button

> The display switches to the incremental DRO view, showing the incremental distance from the start point

The following buttons are available while running a feature:

Option	Function
←	Tap ← to return to the previous pass
→	Tap → to advance to the next pass
End	Tap the **End** button to end execution of the milling operation

The tool radius compensation is applied based on the radius of the current tool. If the plane selection involves the tool axis, the tool tip is assumed to have a ball end.

▶ Move to the start point and make a plunge cut or the first pass across the surface

▶ Tap → to continue with the next step along the contour

> The incremental display shows the distance from the next pass along the line, or arc's contour

▶ To follow the contour, move the two axes in small steps, keeping the X, and Y positions as close to zero (0.0) as possible

> When no step size is specified, the incremental display always shows the distance from the closest point on the arc

The contour view shows the position of the tool relative to the milling surface. When the cross hair representing the tool is on the line representing the surface, the tool is in position. The tool cross hair remains fixed in the center of the graph. As the table is moved, the surface line moves.

▶ Tap the **End** button to exit the milling operation

> ℹ️ The tool offset direction (**R+** or **R-**) is applied based on the tool position. The operator must approach the contour surface from the appropriate direction for tool compensation to be correct.

**7**

Turning specific operations

7.1　Overview

This chapter describes operations and features specific to turning applications.

> **i** Make sure that you have read and understood the "Basic Operation" chapter before performing the activities described in this section.
> **Further information:** "Basic operation", Page 23

7.2　Diameter and radius measurements

Drawings for lathe parts usually give diameter values. The product can display either the diameter or the radius. When the diameter is being displayed, the diameter symbol Ø is shown next to the position value.
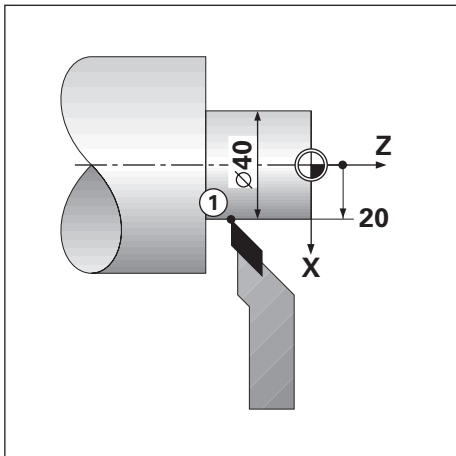
Example:
- Radius display, position 1, X = 20
- Diameter display, position 1, X = Ø 40

Switching between radius and diameter measurements

> **i** The **Rad**/**Dia** toggle is only available when the **Application** is set to **Turning**.

To toggle between radius and diameter measurements:

X
- ▶ Tap the axis button for the axis you would like to change

▶ Tap the **Rad**/**Dia** toggle

7.3　Tool table

The Tool Table is used to store the dimensional offsets for up to 99 tools.

When you change a workpiece and establish a new datum, all tools are automatically referenced from the new datum.

Opening the tool table

To open the tool table:
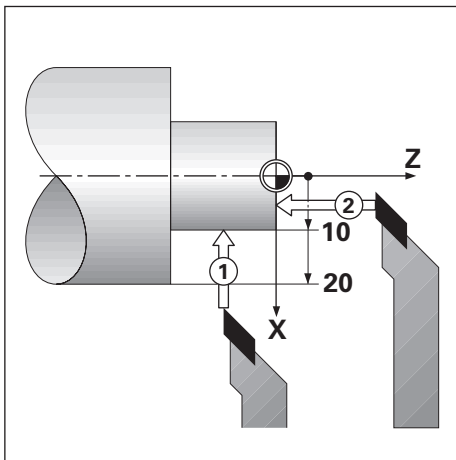
▶ Tap the **Set Tool** button

> The Tool Table opens.

7.3.1 Setting tool offsets

Before you can use a tool, you must enter its offset (the cutting edge position). Tool offsets can be set manually or using the Lock axis feature.

Setting with a known dimension

A tool's offset can be set manually, using the tool, when the diameter of the workpiece is known.

To set a tool offset manually:

▶ Touch the known diameter in the X axis **1**

▶ Tap the **Set Tool** button

▶ Tap **+** to add a new tool, then enter a tool number

or

▶ Tap the desired tool

▶ Enter the position of the tool tip, for example, X=10 in the **X** axis field

> **ℹ** Remember to ensure the product is in diameter display mode Ø if the input is a diameter value.

▶ Touch the workpiece face with the tool **2**

▶ Enter Zero for the position of the tool tip, Z=0 in the **Z** axis field

▶ Tap the **Save** button to save the tool

or

▶ Tap the **Save & Set** button to save the tool and set it as the current tool

or

▶ Tap **Discard** to discard the changes

Setting with an unknown dimension

The Lock Axis feature can be used to set a tool's offset when a tool is under load and the diameter of the workpiece is not known.

The Lock Axis feature is useful when determining tool data by touching the workpiece. To avoid losing the position value when the tool is retracted to measure the workpiece, this value can be stored by tapping the Lock Axis button.
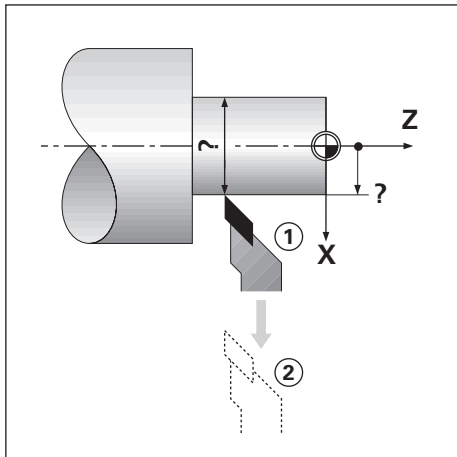
To set a tool offset using Lock Axis:

▶ Tap the **Set Tool** button

▶ Tap **+** to add a new tool, then enter a tool number

   or

▶ Tap the desired tool

▶ Turn a diameter in the X axis

   🔒   ▶ Tap the Lock Axis button while the tool is still cutting

▶ Retract from the current position

▶ Turn the spindle off and measure the workpiece diameter

▶ Enter the measured diameter or radius

   Remember to ensure the product is in diameter display mode Ø if you input a diameter value.

▶ Tap the **Save** button to save the tool

   or

▶ Tap the **Save & Set** button to save the tool and set it as the current tool

   or

▶ Tap **Discard** to discard the changes

7.3.2 Setting a tool

Before you start machining, set the tool you are using from the Tool Table. The product then takes into account the stored tool data when working with tool compensation.

To set a tool:

▶ Tap the **Set Tool** button

▶ Tap the **Set** button next to the tool you would like to use

7.3.3   Deleting a tool

Tools that are no longer needed can be deleted from the Tool Table.

To delete a tool:

▶ Tap the **Set Tool** button

▶ Tap the tool you would like to delete

🗑	▶ Tap the **Trash** icon

▶ Tap **Delete** to confirm deleting the tool

7.3.4   Importing and exporting a tool table

A Tool Table can be imported from an existing file or exported for backup and future use.

Further information: "App Settings", Page 43

7.4   Setting a datum

Datum settings define the relationships between the axis positions and the display values.

For most lathe operations there is only one X-axis datum, the center of the chuck, but it may be helpful to define additional datums for the Z-axis.

The datum table can hold up to 99 datum points.

The recommended way to set datum points is to touch a workpiece at a known diameter, or location, then enter that dimension as the value that the display should be showing.

7.4.1   Setting a datum manually
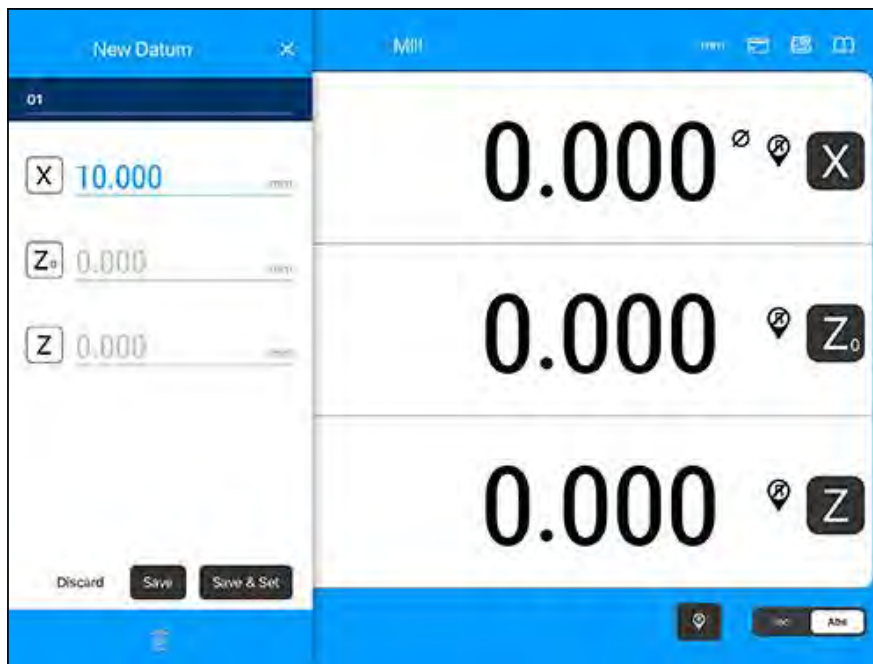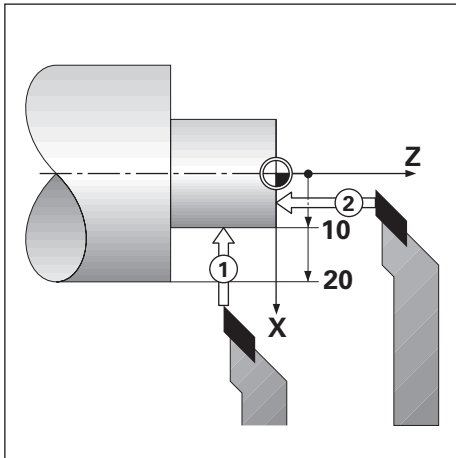
To set a datum manually:

▶ Tap the **Set datum** button
▶ Enter the datum number
▶ Highlight the **X** axis field
▶ Touch the workpiece at point **1**
▶ Enter the radius or diameter of the workpiece at that point

> 🛈 Remember to ensure the product is in diameter display mode Ø if you input a diameter value.

▶ Highlight the **Z** axis field
▶ Touch the workpiece surface at point **2**
▶ Enter the position of the tool tip (Z= 0) for the Z coordinate of the datum
▶ Tap the **Save** button to save the tool

 or

▶ Tap the **Save & Set** button to save the tool and set it as the current tool
or

▶ Tap **Discard** to discard the changes

7.5 Presets

Preset functionality has been explained previously in this manual.

Further information: "Presetting a target position", Page 61

The explanation, and examples on those pages are based on a mill application. The basics of those explanations are the same for turning applications with two exceptions; Tool Diameter Offsets (R+/-), and Radius vs. Diameter inputs.

Tool diameter offsets have no applications with turning tools, so this functionality is not available while doing turning presets.

Input values can be either radius or diameter values. It is important to be sure the units you are entering for the preset agree with the value that the display is currently using. A diameter value is shown with a **Ø** symbol. The value of the display can be changed using the **Rad**/**Dia** toggle, in the axis menu, when in the Turning application.

7.6 Thread Cycle

> ⓘ The **Thread Cycle** feature requires a rotary encoder to be installed to the lathe thread cutting leadscrew.

The **Thread Cycle** feature simplifies and enhances thread cutting on a lathe. The product provides one user definable thread cycle which can be recalled and executed from the DRO at any time during operation, and is remembered when the power is cycled.

Before the **Thread Cycle** feature can be used, **Thread Cycle** settings must be defined.
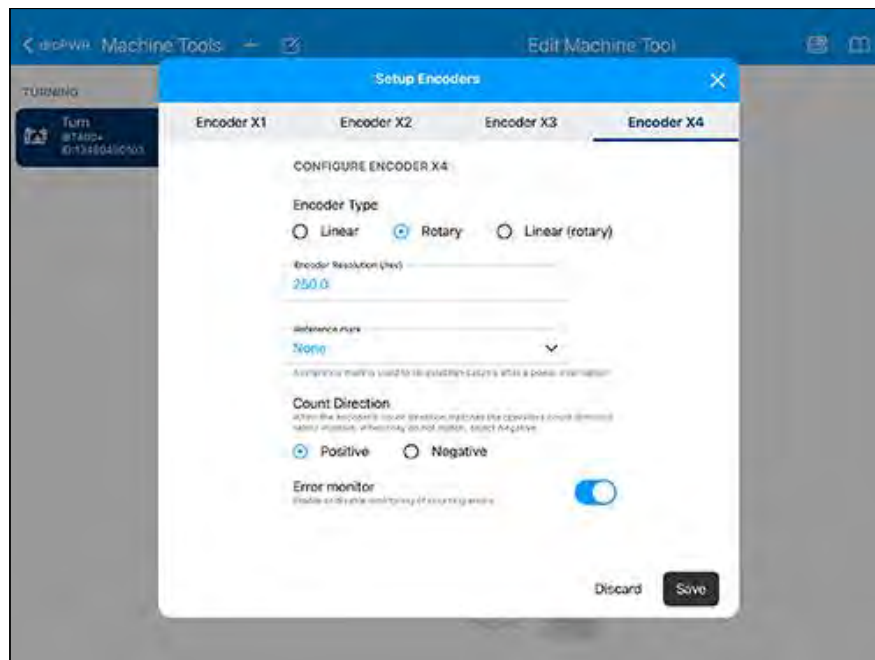
Encoder Setup

> ℹ️ To use the **Thread Cycle** feature on a 3 axes system, the rotary encoder must be installed on the last axis. For a 4 axes system, the encoder must be installed on the third or fourth axis.

> ℹ️ If the current Machine Tool is not setup for the rotary encoder axis a new machine tool will need to be created.

To set up the rotary encoder:

▶ Tap **<**
▶ Tap **Machine Tools**
▶ Tap **Encoders**
▶ Select the axis where the rotary encoder is connected

▶ Select **Rotary** for the **Encoder Type**
▶ Define the remaining encoder setup parameters
▶ Tap the **Save** button to save the settings
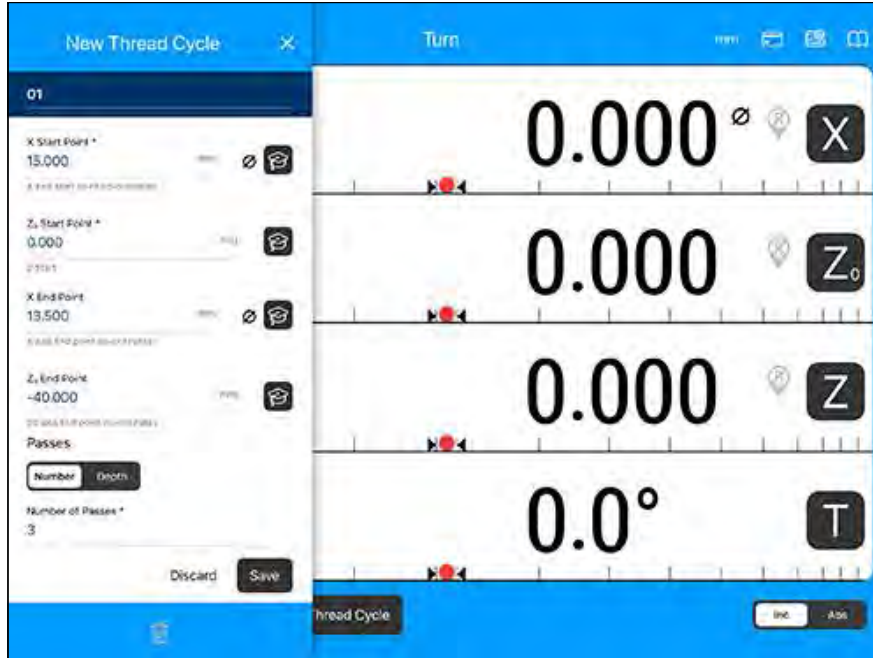
Thread Cycle Setup
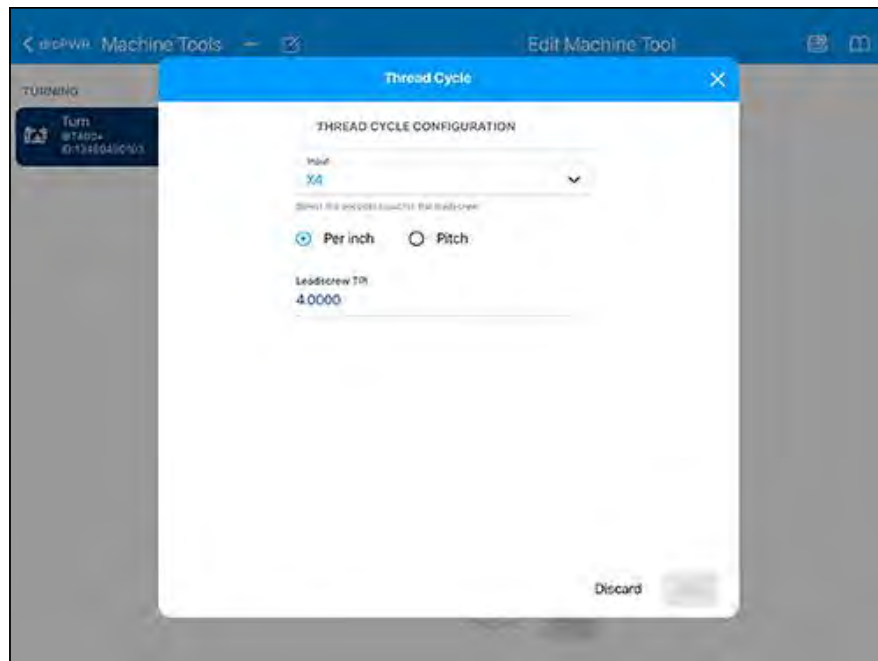
To define the **Thread Cycle** setup:

▶ Tap **<**

▶ Tap **Machine Tools**

▶ Tap the **Thread Cycle** button

▶ Select the **Input** axis that was specified for the rotary encoder

▶ Select thread **Per Inch** or **Pitch**

▶ Enter the leadscrew threads **Per Inch** or leadscrew **Pitch** in the field

▶ Tap the **Save** button to save the settings

Thread Cycle parameters

Once the rotary encoder axis has been defined and the **Thread Cycle** setup is complete, the **Thread Cycle** parameters can be defined.

To define the Thread Cycle parameters:

▶ Tap the **Thread Cycle** button from the DRO screen

> The axes will be in the same mode as what the relative axis is set to: Radius or Diameter

▶ Tap **+** to add a new **Thread Cycle**, then enter a thread number

▶ Enter the **X Start Point** coordinate

▶ Enter the **Z$_0$ Start Point** coordinate

   Typically 0.0 is the normal starting position

▶ Enter the final cut thread diameter in the **X End Point** field

   This is the smaller OD for external threads, and the larger ID for internal threads

▶ Enter the end point of the thread (thread length) in the **Z$_0$ End Point** field

▶ Enter the number of normal passes in the **Number of Passes *** field

   or

▶ Tap the **Number/Depth** toggle and enter the depth of a normal pass in the **Pass Depth *** field

▶ Enter the depth of the **Finish Pass**, or leave blank if no finish pass is desired

> ℹ️ The **Finish Pass** is in addition to the number of normal passes entered, and is included in the total cut depth

▶ Enter the number of threads per inch in the **Threads per inch *** field

   or

▶ Tap the **Per Inch/Pitch** toggle to select **Pitch**

▶ Enter the **Pitch** in mm

▶ Tap the **Thread Side** toggle to select **Inside** or **Outside**

▶ Tap the **Save** button to save the settings

Additional buttons available while in the **Thread Cycle** form:

Button	Function
	Tap the Teach button to set the start and end points

Thread Cycle program
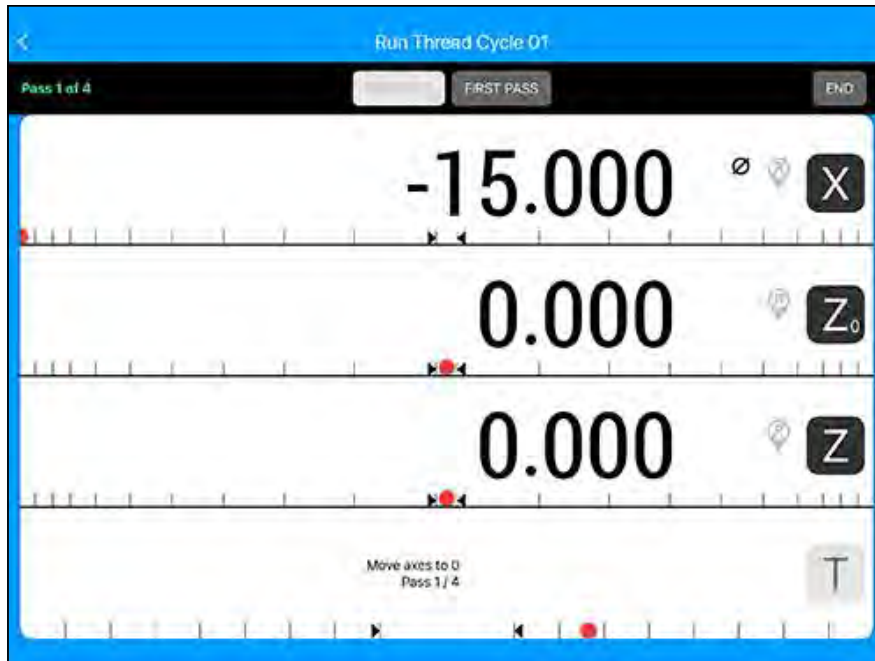
While running a **Thread Cycle**, an instructional message and the pass number display in the Thread Bar display area. The Thread Bar graphic positioning aid displays the rotation of the leadscrew relative to an established mark. This mark is established when the **First Pass** is initially executed. All subsequent passes can now be engaged using the Thread Bar graphic positioning aid when located at the center mark.

The following buttons are available while running a **Thread Cycle** program:

Button	Function
First Pass	Tap the **First Pass** button to start the first pass
Next Pass	Tap the **Next Pass** button to display the next instruction
Start Pass	Tap the **Start Pass** button to start the next pass
Previous	Tap the **Previous** button to restart the procedure
Finish Pass	Tap the **Finish Pass** button to start the last programmed pass
End	Tap the **End** button to exit the program and return to the DRO screen

To run a Thread Cycle program:
▶ Start the spindle
▶ Tap the **Thread Cycle** button
▶ Tap the **Run** button for the **Thread Cycle** you want to run
> The message **Move axes to 0** is displayed
▶ Move the longitudinal axis to zero
▶ Move the cross feed axis to zero
> The message **Press First Pass** is displayed

Running the First Pass
▶ Tap the **First Pass** button
> The message **Ready to engage thread lever** is displayed

> **i** Do not move the carriage manually after tapping the **First Pass** button. Allow the leadscrew to move the carriage so the thread engagement location can be determined accurately.

▶ Watch the machine dial and engage the lever for the first pass on the appropriate number

▶ Keep your hand on the lever while the carriage is moving

> The longitudinal axis will move towards zero

Running the remaining passes

▶ Disengage the lever and back off the cross feed at the same time when the DRO reads 0.0

▶ With the lever disengaged, tap the **Next Pass** button

▶ Move the axes back to the start position. Move Z to 0.0, then move X to 0.0

> The message **Press Start Pass** will display

▶ When ready, tap the **Start Pass** button

> The message **Engage Lever** will display

▶ Watch the Thread Bar Indicator and engage the lever when the indicator turns green

▶ Repeat this procedure until all normal passes are completed

> If a finish pass step was created, the **Finish Pass** button will appear

Running the Finish Pass

▶ Tap the **Finish Pass** button, and execute the same as the previous passes

▶ Tap the **End** button to exit the program and return to the DRO screen

> **i** If a false start occurs at any time, disengage the lever and back off the cross feed at the same time. Tap the **Previous** button to restart the procedure.

HEIDENHAIN CORPORATION

333 East State Parkway
Schaumburg, IL 60173-5337 USA
☎   +1 (847) 490-1191
FAX +1 (847) 490-3931
E-Mail: info@heidenhain.com

www.heidenhain.com