# **Tutorial Manual**

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## TABLE OF CONTENTS

Introduction	5
Technical information on POWERSTATION	5
Technical information on POWERSTATION	5
OBTAINING TECHNICAL SUPPORT	8
Trademark and Copyright Acknowledgements	8
Installation and Setup	.11
Basics	.13
Entity Selection Menu	.13
Entity Masks	.15
Hot Keys and Function Keys	.15
GET POINT MENU	.16
GET DISTANCE MENU	.17
Directory pick lists	.17
Angle Entry	.19
LAYERS	.20
SCALE	.20
Tutorial	.21
MACHINING	.34
MACHINING	.34
DIMENSIONING	.49
DIMENSIONING	.49
MACHINING	.77
MACHINING	.77
Index	.99

### INTRODUCTION

POWERSTATION represents a new direction in CAD/CAM software by breaking some long standing traditions:

- CAD/CAM is much more expensive then general software like word processors, spread sheets and data bases.
- The software has to come from several sources, since no single vendor can make both good CAD and CAM.
- To run CAD/CAM you MUST purchase the latest and most expensive computer hardware.
- Only the "Big shops" need and can afford the cost of CAD/CAM.

We at MICROCIMM felt it was time to shatter these myths:

- With the inexpensive, high quality tools available to a software developer these days, the time required to produce a quality program has been reduced substantially.
- MICROCIMM has brought together experts from both the CAD and CAM fields. And given them a common goal.
- While others say "You need better hardware to run our software" we say "We will make our software better, to run on your hardware."
- At our low prices, even the "One man shop" can benefit from high quality CAD/CAM.

### **TECHNICAL INFORMATION ON POWERSTATION**

POWERSTATION consists of over one Million lines of OBJECT PASCAL. The compiler used is Borland internationals Delphi 5.0/Delphi 7.0, and Embarcadero® Technologies, Inc Delphi 2010.

#### Examples:

The following pages contain some sample POWERSTATION drawings.







### **OBTAINING TECHNICAL SUPPORT**

Come visit us at <u>WWW.MICROCIMM.COM</u> for the latest support information

Email us at SUPPORT@MICROCIMM.COM

Please include:

- Company name
- Your name
- Software product name & version number (Start the program, and select "Help" then "About" for the version name/number)
- Complete detailed description of the problem
- Any files that are involved, such as:
  - Powerstation drawing file (\*.PSD)
  - Powerstation Job file (\*.PSJ)
  - NC-Code file (\*.TAP),(\*.NC)
  - DXF file (\*.DXF)

# *For additional information on support services, please see the brochure that came in your software envelope.*

#### Trademark and Copyright Acknowledgments

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Turbo Power Software Woll to Woll Software

#### HARDWARE REQUIREMENTS

POWERSTATION 2015 was designed from the ground up to support the latest 32/64 bit versions of WINDOWS.

Basically the requirements are that of the installed operating system.

The following are some "practical" minimums:

**CPU** : Intel I3 or higher

Memory: 4 GB

Video : Graphics board and Display Adapter that supports 1024 X 768 resolution or higher.

**Operating System:** WINDOWS XP / 7 / 8 / 10

### INSTALLATION AND SETUP

In order to begin using POWERSTATION it must first be installed on your computer.

# MICROSOFT WINDOWS must be installed and running before POWERSTATION for WINDOWS can be installed.

#### <u>Setup</u>

Insert the POWERSTATION CD disk in the drive. After a few minutes you will be prompted for the default directory for installation\*. The default setting of "C:\PS" should be fine for most all situations. If another directory or drive is desired, type in the full drive / path name at this time.

At this time POWERSTATION will automatically install all of its files, and create the proper start menu items for launching POWERSTATION and its associated programs. The default is under the group called "MICROCIMM".

\* If the setup does not begin automatically, start an explorer window by holding the Windows key and pressing the letter "E". Double click on the CD drives icon, and finally double click on the file "SETUP.EXE"

#### **Entity Selection Menu**

During the operation of POWERSTATION, frequently you will be asked to select the desired entity or entities (point, line, arc, etc.). For easy identification selected entities are changed from their current color and line style to a white color and a dashed line style. Generally you will be presented with the entity selection menu:

		Done/ <escape></escape>
Command	"Hot Key"	All Entities
All Entities	A	By-Layer
By-Layer	В	By-Type
By-Type	т	Chain
Chain	С	Chain
Single	S	Single
Window	W	Window
Last	L	Last
Remove	R	Permana
Z-Level	Z	Kemove
Arc-Radius	D	<u>Z</u> -Level
Line-Length	N	Arc-Ra <u>d</u> ius
By-Color	Y	Line-Length
Outside	0	- <u>-</u>
		RT-Color
ALL Selects every entity in the	current drawing.	Outside

**BY-LAYER** -- Prompts you to select a layer from a drop down list, and selects all entities on the specified layer.

Layer Selection	×
Layer Number / Description	
1 - Dimensions	
OK Cancel <u>H</u> elp	

**BY-TYPE** -- Asks you to select a sample entity (Point, Line, Arc, Etc.) the automatically selects all entities of the selected type.

**SINGLE** -- Used to select individual entities, one at a time. The standard mouse cursor (+) will change to a small box shape. When picking an entity, position the box so that part or all of the desired entity is contained inside the box, then press the Left mouse button. After selecting all of the desired entities, press the right mouse button.

**CHAIN** -- The chain selection is used for rapid selection of connected entities that form a uninterrupted path. Note that lines and arcs are the only type of entities that can be chained.

Chaining works in the following manner:

- 1. First you will be asked to select the starting line or arc. The chaining will start at the opposite end of the entity you select. If an entity is found who's beginning or ending point is near (less then 0.0005 away (this number can be changed in the "FILE-PROJECT" section)) from the end of the current entity, it will also be selected. A small arrow will be drawn to show which direction the chaining went in.
- 2. This procedure will continue until no entities are found near the end of the current entity.

**WINDOW** -- Used to automatically select all entities inside of the selected window. You will be prompted to indicate (with the cursor) the two corners of the window.

**LAST** -- Easy selection of the last entity created. Not valid after and MODIFY, or MACHINING commands.

**REMOVE** -- Works exactly like the SINGLE selection above, except that the entities you pick are DE-selected (removed from the selection set).

**OUTSIDE** -- Works as the exact opposite of "WINDOW", selecting only entities who's end points fall outside of the specified window.

**BY-COLOR** -- Works exactly the same as "BY-LAYER". After selecting the color, all entities of that color will be selected.



**ARC-RADIUS** -- After prompting for a Radius value, Every arc with the specified radius will be selected. A good example of using this command would be to select all holes of a certain radius for drilling.

**LINE-LENGTH** -- By specifying the minimum / maximum length of lines to be selected. This command can also be used to select lines that extend way out of the drawing area. For example Shorter than 99999.0, and longer than 50.0 would remove any lines longer that 50 inches. You will be asked:

Find Lines with Length Shorter Than [10.0000] ? Find Lines with Length Longer Than [0.0000] ? **Z-LEVEL** -- Allows entities to be selected based on a -Z- level range that they fall within. This can be useful for modifying (erasing, moving, changing layers) wireframe geometry imported from other CAD systems.

Maximum -Z- Value [0.0000] ? / Minimum -Z- Value [-1.0000] ?

#### <u>Entity Masks</u>

At times the process of selecting an entity with a mouse may not be specific enough to return the desired entity. Example: Suppose you are trying to select a point that is the tangent point between a line and an arc. Simply placing the pick box over the point may select the line, arc, or point, depending on which entity is internally stored first. However by setting the proper mask, your cursor selection will be limited to the type of entity selected in the mask. To change the current mask, simply select a new Mask from the tool bar on the top of the display area (In the picture that follows, the mask box is the one that reads "All Entities"). IT IS A GOOD IDEA TO RETURN THE MASK SETTING BACK TO "ALL ENTITIES" AFTER THE CURRENT COMMAND. FORGETTING TO DO THIS CAN MAKE SUBSEQUENT COMMANDS APPEAR TO MALFUNCTION.



#### Hot Keys and Function Keys

Hot keys and function keys are used for speedy selection of commands. These commands are generally available all of the time with the exception of while you are answering a prompt.

<f1></f1>	Help system
<f5></f5>	Display Refresh
<f7></f7>	Delete Last
<f8></f8>	Current Color
<f9></f9>	Layer Control Dialog
<ins></ins>	Repeats the last command (menu selection)
<ctrl>0</ctrl>	Jumps directly into the "operations" Dialog.
<ctrl>P</ctrl>	Print
<ctrl>C</ctrl>	Copies the current image into the WINDOWS clipboard
<ctrl>N</ctrl>	Run the Post processor
<ctrl>E</ctrl>	Zoom Extents
<ctrl>R</ctrl>	Zoom Previous
<ctrl>Z</ctrl>	Zoom
<ctrl)v< td=""><td>Starts the Solid Verification (PRO/PRO+ Only)</td></ctrl)v<>	Starts the Solid Verification (PRO/PRO+ Only)

Additional Hot keys will be described in the "GET POINT", "GET DISTANCE", and "ENTITY SELECTION" text.

### **GET POINT MENU**

Within most all commands that require an X and Y coordinate you will be presented with the "GET POINT" menu. This menu provides a great deal of flexibility in specifying an X and Y location.

The menu displays:

Done/<	Escape>
<u>C</u> enter	
Coo <u>r</u> dir	iates
Cur <u>s</u> or	
<u>D</u> istanc	e From End
End-Of	
<u>Intersec</u>	tion
<u>L</u> ast	
Mid-Po	int
On-An-	Arc
Point	
<u>N</u> ear	
Origin (	X/Y <u>Z</u> ero)

<b>Selection</b>	<u>Hot</u>	Description
<b>C</b>	<u>Key</u>	
Center	С	Center point of an Arc or Ellipse
Coordinates	R	Prompts for the absolute X and Y coordinates.
Cursor	S	Displays a cross cursor for input of a screen position.
End-Of	E	Returns the End point of a Line, Arc, or Ellipse that is closest
		to the selection position.
Intersection	Ι	Finds the intersection of any combination of lines and arcs.
Last	L	Returns the X and Y location of the last screen location
		selected.
Mid-Point	Μ	Finds the Mid-point of a Line or Arc.
<b>On-An-Arc</b>	0	Calculates a point on an Arc or Ellipse, at a specified angle.
Point	Р	Returns the X and Y location of a previously defined point.
Near	Ν	Calculates a point on an Arc or line near the selection point.
Origin (X/Y Zero)	Ζ	Enters the default origin of 0,0
Obround	В	Specifies the center of an obround shape (Power-fab only)

#### **GET DISTANCE MENU**

Within most all commands that require an X and Y distance you will be presented with the "GET DISTANCE" menu. This menu provides a great deal of flexibility in specifying an X and Y distance.

Done/ <escape></escape>
Coo <u>r</u> dinates
Cur <u>s</u> or
<u>D</u> elta-Point
<u>T</u> wo-Points
<u>P</u> olar Coords

<u>Selection</u> Coordinates Cursor	<u>Hot Key</u> R S	<b>Description</b> Prompts for the absolute X and Y distances. Displays a cross cursor for input of two screen positions then calculates the true distance between
Delta-Point	F	these two positions. Prompts for a point using the "GET POINT" menu, then asks for an X then Y distance. The X and Y distance will be added to the selected point to calculate the total distance
Two-Points	Т	Prompts for two points using the "GET POINT" menu then calculates the difference between them.
Polar Coords	Р	Prompts for a base point using the "GET POINT" menu. Additionally you will be prompted for the distance and angle from the base point.

#### Using the POWERSTATION HELP System

POWERSTATION – Learning to use the on-line help systems cannot be stressed enough, all of the information in the help files can be searched much faster then text based manuals, and we expect to be constantly updating the on-line help, while keeping manual revisions to a minimum. If you are not familiar with the features of the WINDOWS help system, in POWERSTATION select "Help", then "Contents" to navigate through the various topic. "Help Search for Help on.." and type in the name of a topic. Pay special attention to the "Search" TAB, it can help you to find information just by knowing one of the key words involved, example; typing "DXF" in the "Search" page, and several areas of the manual where DXF files are discussed.

# The on-line help files contain the detailed reference on every command / feature in POWERSTATION, this manual is intended primarily as an introduction/tutorial.

#### 18 **POWERSTATION CALCULATED FIELDS**

To simplify the entry of numbers, the standard Windows control "Text Box" has been extended to allow the entry of numbers as calculated expression.

The calculator function works as a twenty two function scientific calculator and is available any time throughout POWERSTATION whenever a number must be input. Please Note: All trigonometric functions are in radians.

Sin Sine	Cos Cosine	Tan Tangent
Sec Secant	CSC Cosecant	Cot Cotangent
Asin Arc Sine	Acos Arc Cosine	Atan Arc Tang
Sinh Hyperbolic Sine	Cosh Hyperbolic Cosine	Tanh Hyperbolic Tangent
Sech Hyperbolic Secant	Csch Hyperbolic Cosecant	Coth Hyperbolic
Ln Natural Log	Log Log Base 10	<b>PI</b> 3.14159
Abs Absolute Value	Sqrt Square Root	Exp Exponential
Fact Factorial	Sqr Square (X*X)	

Rd Constant to convert Radians to Degrees 57.29578

Dr Constant to convert Degrees to radians 0.01745

**Ft** Constant to convert feet to inches 12.0

% Character to convert results from metric to inch mode Standard Operator's (in order of precedence):

- ^ Exponential
- \* Multiplication
- / Division
- + Addition
- Subtraction

Parentheses '()', Square Brackets '[]', and Curly Brackets '{} 'may be used to override the normal order of evaluation.

SIN(23DR)	(sine of 23 degrees)
Sqrt(SQR(3.5)+SQR(2))	(Square root of 3.5 squared plus 2 squared)
2 + (23/64)875	(2 plus 23/64 minus .875)
SIN(10+23/60+13/3600DR)	(Sine of 10 degrees23 minutes and 13 seconds)

End Point of Line	×
Distance :	23/640025
Angle :	0.0
OK (	Cancel <u>H</u> elp

Notice that the distance has been entered as "23/64-.0025". When the input is moved to the next "text box" by pressing the <Tab> key or selecting a new box with the mouse, the "Distance" box will be changed to read "0.356875".

#### ENTERING DIAMETERS

When prompted for a radius dimension, diameters can be enter simply by adding the letter "D" Example:

#### Enter the Radius [2.375D]

The radius value will be recorded as 2.375/2 or 1.1875.

#### <u>Angle Entry</u>

In POWERSTATION angles are measured in degrees from the three O'clock position. Counter clockwise angles are specified as positive values, and clockwise angles are specified as negative values. In other words +45 and -315 represent the same angle. If minutes and seconds are to be entered you enter them directly in the dialog. Example:

For 30 degrees,	12 minutes,	16 seconds	type:
-----------------	-------------	------------	-------

Enter the Angle	×
90  -270 -180  -270 -360  270 -90	Angle :0.00000Degrees :0Minutes:0Seconds :0
ОК	Cancel <u>H</u> elp

#### LAYERS

In a CAD/CAM system layers can be thought of as clear overlays that can be placed over the current drawing and removed at will. POWERSTATION supports a full 256 layers, each with it's own user selectable color, line style, and description, to allow a great deal of flexibility. Layers are commonly used to separate parts of the drawing which may "clutter up" the screen. Typically one might use one layer for the basic part shape, one layer for dimensions, one for the title block, one for each operation of the machining cycle. Each layer can be individually turned ON or OFF. When a layer is turned OFF any information stored on it is not drawn on the display screen and cannot be selected with the mouse (although it still exists in the drawing).

#### <u>SCALE</u>

For newcomers to CAD/CAM the drawing size and scale can be confusing. In traditional manual drafting, you must choose a scale at the start and everything you do revolves around this scale. In CAD/CAM the scale seen when viewing or plotting is insignificant, as it can be changed at any time with simple commands. A few point to remember:

- 1) It is generally a good practice to draw everything at its natural 1:1 scale, and use the "ZOOM" family of commands to change what is displayed on the screen.
- 2) Remember that the scale that he drawing is displayed at has no affect on the part coordinates. The only "scale" commands that in any way change the part coordinates is "BLOCK-MERGE-SCALE", and "MODIFY-SCALE".
- 3) No matter how large or small a part is it can always be blown-up to fit the entire screen (or plotted sheet).
- 4) No matter how small a detail is, you can "ZOOM-IN" until it is visible.

#### Additional Information

The libraries and bookstores contain a many volumes on CAD/CAM techniques. Although the exact commands used in other CAD systems may be different, the principals used to create drawings will be the same. These books will typically go much more into detail on the CAD drafting aspects then this manual.





Figure 4-0 The sample drawing

The purpose of the tutorial is to guide you step by step through the creation of a POWERSTATION drawing. The drawing shown in figure 4-0 will be taken from the initial geometry creation, to machining, and finally dimensioned and plotted. Each step will be numbered for easy reference. The information you are required to type will be underlined. There are two tutorials, one for milling, and one for turning.

#### <u>STEP 1</u>

We begin by starting POWERSTATION. To do this: Press [Start] (on the WINDOWS Tool bar), then select "Programs", then "MICROCIMM", then "POWERSTATION"



Figure 4-1 The opening screen

#### <u>STEP 2</u>

At this point the POWERSTATION opening screen will be displayed (Figure 4-1). We will begin by drawing a rectangular box which will be used as the basic shape of the part.

To create the box we select, from the main menu select "Geometry", "Shapes", "Box"

ntry Method	Coordinates	DUX
Specify Width & Height		
C. Constitu Consord Corpor	Width:  0.0000	
specity second corner	Heigth: 4 00000	
Enter Coordinates>	101gth.  4.00000	Box Width
	Base -X- : 0 0000	the the
ase Point Defines	1	
Corner Point of Box	Base -Y- : 0.00000	Box Height
Center Point of Box	1	A Radius/Chamfer Size
Corner Shape		1 1
Sharp C Radius	Chamfer	
Boy Corner Badius/Chamfer	0	Base Anale
	U	Duse migre
Base Angle : 0.	0	1×7
		— Base Point of Rotation
		Same as First Point
		(Corner or Center)

At this point the above dialog will be displayed. For this example we need to change the width to 5.5 and the height to 4.0. Press [OK] with the mouse.

At this point the screen will look like figure 4-2.

Eile	WERST	ON 2015 [Untitled] – O etry Mgdify Drafting Machining Display Help	×
	•	🙀 💾 🕫 ♀ 🕞 All Entities – Select 🔄 💡 📗 📗 🔍 🍳 🔩 💯 🧾 🦎 📯 '- View 🚽 🔶	
<b> </b>   <b>×8</b> .	5723	/-5.1661 Z0.00000   Layer 0	
R	ā,		
8	P		
\$₽	0		
Q			
₿.	A		
₽₽	6		
×	°°°		
<b></b>	0-0 <sup>-0</sup>		
0P#20 0P#30			
ľ	Q		
N 100 N201	888		
			-
Copyri	aht © 19	5000 y 1000 Software_Ltd. MOD No Op# Tool: X 0.00000 Y 0.00000 Z 0.00000	

Figure 4-2 the basic part shape (box)

#### <u>STEP 4</u>

It will be easier to work if the box were more centered in the display screen. To do this we will use the "**Display-Zoom Extents**" menu command (or press <ctrl>E).

Eile F	WERST	ATION 2015 [Untitled]	- 0	×
B	<b>B</b>	월 월 🎽 🗠 🖓 All Entities - Select 🚽 ? 📕 📗 🔍 역, 역, 1월 💹 🖹 년 XY-View - 🔶		
X7.	5091	Y0.6934 Z0.00000 Layer 0 戦) ロロ ③ 〇 メ ☆ ③ タ		
N N N	₹\$ <b>₽</b>			-
0P#10 0P#20 0P#30 0P#30 N150 N150 N150				-
				·
Copyrig	ht © 198			

Figure 4-3 The display has been "Zoomed"

Figure 4-3 shows the new box location. Note that the fitting may change the position of the box on the display but not its size or location in space.

#### <u>STEP 5</u>

Looking ahead, we only need a small section of the line on the top. The remainder of the line will intersect with several other parts of the final geometry adding unnecessary difficulty to the clean-up "Trimming". To do this we will change the end-point on the left side. Select "Modify", "Trimming", "Change Line End".



Next, the Get Point menu (See chapter Basics) will be displayed. Select "Coordinates" from the menu. Enter in coordinates of X5.0, Y4.0.

-X- Coordinate	
5.00000	
-Y- Coordinate	
4.00000	
Enter Point Coordinates	
Enter Point Coordinates	
 Enter Point Coordinates	



#### <u>STEP 6</u>

Next we will enter the 1 3/8" arc. Select "Geometry", "Arc", "Coordinates" from the main menu. You will be asked for the coordinates:

1.25000       1+3/8         -Y- Coordinate       Start Angle         3.00000       0.00000         -Z- Coordinate       End Angle         0.00000       360.00000	-X- Coordinate	Arc Radius
Y- Coordinate Start Angle 3.00000 Z- Coordinate End Angle 0.00000 360.00000	1.25000	1+3/8
3.00000 Z- Coordinate D.00000 360.00000	Y- Coordinate	Start Angle
Z- Coordinate End Angle 0.00000 360.00000	3.00000	0.00000
0.00000 360.00000	Z- Coordinate	End Angle
	0.00000	360.00000
Enter Arc Coordinates	Ente	r Arc Coordinates



Zoom Extents again and the display should now look like Figure 4-4.

Figure 4-4 The first (1 3/8") arc

#### <u>STEP 7</u>

Now lets create the line tangent to the 1 3/8" arc. First select "Geometry", "Line", "Tangent to One" from the menu.

You will now be asked to "Select a Point or ARC". You should now select the arc, anywhere on its upper right side. See figure 4-5.



Figure 4-5 Creating the tangent line

After picking the approximate tangent location, you will be asked:



Notice the wrong line has been created (see figure 4-5). Pressing the  $\Leftarrow$  icon on the tool bar **(Undo)** will remove it. This is a good time to see the effects of using the  $\Leftarrow$ Undo and  $\Rightarrow$ Redo commands. Repeat the steps to create the line, but use -45 for the angle. See figure 4-6



Figure 4-6 The corrected tangent line

#### <u>STEP 8</u>

Now we can create the line. Select "Geometry", "Line", "Tangent to One" from the menu. You will now be asked **"Select a Point or Arc, or <Escape> for Point Menu"**. Press the <Esc> key or click the right mouse button and select <Escape> from the menu.

The Get Point menu (See chapter Basics) will be displayed, select "End Of", then select the left end of the line.



#### <u>STEP 9</u>

We need one more piece of geometry, the 1/2" radius on the top. To add this we will save a few steps and use the "**FILLET**" command, which will not only add the fillet, but will clean-up (Trim) the lines being filleted.

Select "Geometry", "Arc", "Fillet" from the menu. You will now be asked:

First Item - Select a Point, Line or Arc Second Item - Select a Point, Line or Arc Enter the Radius [1.3750] ? <u>.5<Enter></u>

Note: Normally the entities to be filleted must be selected in a counter-clockwise order, or the wrong fillet will be generated. Note: This is not generally necessary when filleting lines, the system will generate the proper fillet automatically (This is NOT true when filleting lines to arcs, or arcs to arcs, the counter-clockwise picking is required).



#### <u>STEP 10</u>

For Trimming, we will be using the "Modify", "Trimming", "EASY-TRIM" command. After selecting this command you would simply point (with the mouse) to the sections of lines or arcs that you want to remove. Simply click (In Order) on the points in the following examples:





Figure 4-19 The part after trimming

### <u>STEP 11</u>

We will draw the drilled holes, using a "linear pattern'.

From the menu, select "Geometry","Pattern","Linear. You will be asked for the basics:

Patter	rn Information
	Create Pattern Of Holes Radius: 0.125 Points
	-Z- Level of Pattern : 0.00000
	OK Cancel <u>H</u> elp

Next you will be asked to enter the start point of the pattern:

Done/ <escape></escape>	Done/ <escape></escape>									
<u>C</u> enter										
Coo <u>r</u> dinates										
Cur <u>s</u> or										
Distance From End										
<u>E</u> nd-Of										
Intersection										
<u>L</u> ast										
<u>M</u> id-Point										
<u>O</u> n-An-Arc										
<u>P</u> oint										
<u>N</u> ear										
Origin (X/Y <u>Z</u> ero)										

Select "Coordinates", and enter the following:

Coordinate Entry		×
-X- Coordinate 1.00000		
-Y- Coordinate 1.00000		
	Enter Point Coordinates	
<u>ок</u>	Cancel <u>H</u> elp	

Next we enter the pattern details:

Linear Pattern	×
Number of Holes : 4 End Point of Pattern is Known Distance / Angle Distance Between Holes : 1.0	Linear Distance Between End Point Start Point
Base Angle of Holes : 0.0	Base Angle



The screen will now look like this

Note: This step is purely for drawing purposes, if all that you want to do is drill these holes, this step could be skipped.

#### <u>STEP 12</u>

At this point the part geometry has been generated and it is a good time to save the work done so far. You should save your work every few minutes, so if anything goes wrong (power-out, disk fails, etc.) not all of your work will be lost.

To save the drawing, select "File", then "Save" from the menu. At this point you will be asked:

Enter LESSON1 as the file name, and press [Save]. (Notice, that since the file has never been saved, you were automatically given the "Save As" dialog.



#### <u>STEP 13</u>

### MACHINING

POWERSTATION uses a method of operation modeled after the way a machinist thinks. We call this method "OPERATION ORIENTED". NOTE: POWER-CAD users, please skip ahead to Dimensioning).

The first step is to define one more operations that will take place. The second step is to select the operation that you want to work on, and finally do the machining for that operation.

Note that the operations must be specified in the order that you want the part machined in, but they do not have to be selected & worked on in any specific order. POWERSTATION will automatically generate the operations in the originally defined sequence no matter what order the operations were selected in.

If you make a mistake while doing any machining operations, first try pressing the <F7> (DELETE-LAST) key. If this does not produce the desired results, simply select the "Machining", "Tool Path|Edit", "Delete-Operation" command to delete all tool motion from the current operation, re-select the operation and try again.

Select "**Machining**", "**Operations**" from the menu (or press the operations ICON on the icon bar (located on the left side of the screen). Not sure which is the "operations" icon ? Simply hold the mouse cursor over the icon in about two seconds, a "Hint" will be displayed describing the function of the icon.

Machining Operations Manager	- 0	×
Operation List Details	Material Tooling Tool Shape	
Current Operation # Post	Color Description	^
		~

Figure 4-21a (The Machining Operations Manager)

To enter an operation:

- 1) Click the cursor in the "Op #" column, and type in the number of the operation.
- 2) Double click the mouse on the "Color" column, and select the desired color.
- 3) Click the cursor on the "Description" column, and enter a description of the operation
- 4) Double click on the first "Current" column, and the word "Yes" will appear. This indicates that this is now the currently selected operation.
- 5) At this point the dialog should look like figure 4-21b
- 6) Click on the tab that reads "Details". This will switch to the "details" page where you enter the specifics on the currently selected operation (tool number, diameter, etc..). See figure 4-21c

Machin Operation	ning Operation	ns Ma	anager										-	×
				5	<b>/1</b> 0									
Opera	ation <u>L</u> ist	Det	tails	Mate	rial	Tooling	Tool	<u>S</u> hape						
Current	Operation	. #	Post	Color	Desc	ription								^
Yes	10		X		ROUG	H PROFILE	.75	END MII	LL	 	 		 	
. <mark>.</mark>										 	 	 	 	 
														~

Figure 4-21b (Basic operation information)

Machining Operations Ma	nager			-	×
Operation List De	etails Material Tooling	g Tool <u>S</u> hape			
Operation #	Operation Type Milling 🗸	Tool Number	Offset Number		
Fixture Offset	Tool Diameter	Stock Allowance	Corner Radius		
Feed © IPM C IPR 8.00000 Description:	Spindle / © RPM C ( V Forward	Speed CSS 2000 Range #1 V	Coolant : Flood 💽		
ROUGH PROFILE .75	6 END MILL		Set Home		
			Set <u>G</u> age		

*Figure 4-21c (Operation #10 Details)* 

Next define the remaining operations, using the information displayed in figures 4-21d,e, & f.

Machin	ing Operation	ns M	anager									-	×
<b>B</b>		1	1	5	<i>0</i> <b>1</b> 0								
Opera	tion List	De	tails	Mate	rial	Tooling	Tool	L <u>S</u> hape					
Current	Operation	. #	Post	Color	Desc	ription							 ^
Yes	10		X		ROUG	H PROFIL	E .75	END MILL					
	20		X		FINI	SH PROFI	LE .5	END MILL					 
	30		X		1/4	DRILL							
			[										
			ļ										 
					ļ								 
	; ; ; ; ;		<u>.</u>										 
			+		+								 
			+		1 								 
			1	Fior	iro 4	1-21d (	411 0	nerations	for the t	itorial par	<i>t</i> )		
📕 Machin	ning Operation	ns M	anager	1 150		214 (1	111 0	peraitons	jor me n	nor iai par	/	-	×
Operation	Help		~	<b>m</b> 1	. 6.	1							
		1				]		1					
Operat	ion <u>L</u> ist	D	etails	Mate	rial	Tooling	Tool	L <u>S</u> hape					
Opera	ation #		C	perat	ion I	ype		Tool Number	- 	Offset Numb	er I		
Fivtur	a Offeat				Diama	⊥ tar	q	12 Lock Allows		Corper Padi			
0	•			0.50	000		5	0.00000		0.00000			
Feed					Sp	indle / S	Speed			Coolant :			
• IP	G IPR				(° I	RPM C C	ss 200	00		Flood -			
8.00	0000				•	Forward	Range	e #1 ▼					
Desci	ription:												
FINI	SH PROFILI	Ξ.	5 END	MILL							Set Home	1	
												1	
											Set <u>G</u> age		

*Figure 4-21e (Operation #20 for the tutorial part)*
#### Mill Tutorial 37

Machining Operation	is Manager	- 🗆 X
Operation List	Details Material Tooling Tool Shape	
Operation #	Operation Type Tool Number	Offset Number
Fixture Offset	Tool Diameter Stock Allowance	Corner Radius
Feed © IPM © IPR 8.00000	Spindle / Speed • RPM C CSS 2000 • Forward Range #1	Coolant : Flood 🗸
Description:	,	
1/4 DRILL		Set Home
		Set <u>G</u> age

*Figure 4-21f (Operation #30 for the tutorial part)* 

## <u>STEP 14</u>

At this time it is a good idea to make a print out of the defined operations. To do this select "Operation", "Print" from the menu. Next we need to make operation #10 active. To do this, double click on the "Current" column for operation #10, then exit from the Machining Operations Manager.

### STEP 15 (Optional)

It is a good practice to place your machining on a different layer than the part geometry. While we are dimensioning the part we really do not need to see the tool path, so placing it on its own layer gives us an easy way to "hide" the tool path.

To change the current layer to 100, press the  $\langle F9 \rangle$  key (or select "Display", "Layer-Control" from the menu), then click on "Current Layer" box and type: <u>100</u>, click the cursor on the description box (The right most column in the layer 100 row), and enter in a description like "Machining". Next press the [OK] button.

Num	ΟN	Color	Used	Style	Description 🔺
98	X			Solid	All On
99	X			Solid	
100	X			Solid	Machining All Off
101	X			Solid	
102	X			Solid	Show Used
103	X			Solid	
104	X			Solid	
105	X			Solid	Current Layer
106	X			Solid	· · · · · · · · · · · · · · · · · · ·
107	X			Solid	100 🜩
108	X			Solid	
109	X			Solid	Line Style
110	X			Solid	
					· · · · · · · · · · · · · · · · · · ·

#### 38 POWERSTATION

### <u>STEP 16</u>

To be safe let's save the file again. This time, just press the right mouse button with the cursor anywhere in the main display area. A short menu will popup, Select "Save File".

#### <u>STEP 17</u>

Now we can start rough machining the profile. Before any machining can be done we must select an operation. This should have been done at the end of step 14, but just in case you missed it, select "Machining", "Operations". Double click the mouse in the "Current" column of operation #10, then select "Operation", "Exit" (Or press the exit ICON).

#### <u>STEP 18</u>

Now we will use "chain selection" to mill the outside profile. Select "Machining", "Chain Cutting" from the menu. We will be asked:

```
Select starting line/arc (figure 4-24)
```

Select the Line or Arc to stop before, <Escape> for none (Press the <Esc> key, or click the right mouse button, and select <Escape>)



Press the [Yes] Button (figure 4-25)

```
Pick the side to offset to (figure 4-26)
```



Chain Cut Boundary	X
Chain Cut Boundary         Multiple -Z- Depths       Cutter Comp         Multiple -Z- Depths       Cutter Compensation         Cutter Compensation       Cutter Left         Cutter Right       Automatic         Compensation Number :       1         Image: Turn Off Compensation at End       Image: Compensation at End         General       Type :         Type :       Milling         Image: Cutting Feed :       9.00000         -Z- Cutting Feed :       9.00000         -Z- Clearance Plane :       0.1         Total -Z- Depth :       0.75000	General Multiple 2- Cutter Comp Multiple Stock Approach/Depart EDM
General       Options         Type : Milling       Image: Provide the state of the sta	If Cutter Compensation is ON An Approach & Depart Must be defined!

Enter the information (-Z- feed, clearance, and total depth, etc..) as displayed above. Note; Depending on your preference, you can either leave the "Depart" settings to their default of "none", or set them the same as the "Approach" settings. The display will look like figure 4-27



Figure 4-24 Selecting the starting line or arc



Figure 4-25 The entities have been chained



Figure 4-26 Picking the side to offset to



Figure 4-27 The first operation completed

# <u>STEP 19</u>

Select operation #20 by selecting "Machining", "Operations" from the menu (or clicking the "Operations" icon). Double click on the "Current" column, in the row for operation #20. Exit the Machining Operations Manager.

Next repeat step 18. When done the display will now look like figure 4-28.

**Note:** To finish with the same tool, do not select a new operation (#20), simply select "Machining", "Tool Path Edit", "Stock Allowance", and change the stock allowance to 0.0 before repeating step 18.



Figure 4-28 The second operation completed

### <u>STEP 20</u>

Now for the drilling. Select operation #30 by selecting "Machining", "Operations" from the menu (or clicking the "Operations" icon). Double click on the "Current" column, in the row for operation #30. Exit the Machining Operations Manager.

Select "Machining", "Drilling", "Linear" from the menu.

Linear Pattern          Number of Holes :       4         End Point of Pattern is Known         Distance / Angle         Distance Between Holes :       1.0         Base Angle of Holes :       0.0	Distance Between End Point Start Point Base Angle
OK Cancel Help	

Fill in the "Linear Pattern" dialog as shown above.

Next you be given the "GET POINT" menu, (See chapter "basics" for more information on this menu). Select "**Center**" from the popup menu, then move the pick box on to the first hole and press the left mouse button.

# STEP 20 continued ....

Next you are asked to select the type of drilling cycle, and the drilling depth. Select "Through" from the "Depth Specifier" list, and enter -.75 as the "Depth to Drill Through".

ling Cycle / Depth Selection	1	
)rill Cycle Type	Drill Depth	Drill Depth
Standard Drilling 🔹	Rapid Plane (-R-)	SPECIFYING DRILL DEPTH
Deep Drill Peck Increment	Dill Tip Angle :	
Tapping TPI: 16.0	-Z- Top of Part: Specify Depth By: 0.00000 Through	
Pitch : 0.0625	Depth to Drill Through : -0.75000	
Dwell Time 0.0	Note: Only Center Drills Large / Small Enough to generate the Hole Diameter Speficied Above Will be Shown	AB5 -Z-
Defaults OK	Cancel <u>H</u> elp	



# Completed Drill Cycle

#### 44 POWERSTATION

### <u>STEP 21</u>

Once again, let's save the file as in step 15. After that, let's check the machining status. Select "Machining", "Machining Status". When done viewing the status, press the [OK] button.

#### <u>STEP 22</u>

The final step in machining is to run the post processor. Select "Machining", "Post (Generate NC-Code)" from the menu (or select the Post icon).

From the list of machines in the upper left hand corner of the dialog box, move the highlight to "DMILL.MCH".

Next we will set the name of the NC-tape file to be created. Press the button labeled [Set NC-Code File Name]. Enter "LESSON1" as the "File Name", and press [OK].

Press the [Go] button. The post processing now begins, and may take a few minutes. When done, press [Close] to leave the post processor.

*NOTE:* You will need to add your own machine names to the post processor list. For information on add them, select "Help", "Post Processor Setup".

Machine Name	NC-Code File Name :
Dedm.mch	lesson1.TAP
DLATHE.MCH	Set NC-Code File Name
	Line Number: 302
NC-Code	
N560 M09	~
N570 G28 Z0 M29	
N580 M30	
8	~
<	>

Now that the tutorial part is finished, lets see how the new "Job Processing" feature can help with making changes.

### <u>Step 23</u>

Select "Machining-Job Processing". The screen layout will change and following dialog will be displayed:



# <u>Step 24</u>

In the Job Processing Dialog, in the "Tree area" (Upper left) one at a time, click on the Plus "+" signs. The dialog will now look as follows:



### <u>Step 25</u>

For this example, we will make three common changes and show how quickly we can regenerate the tool path and the NC-Code. For this example, we will say that operation #10 which is currently a  $\frac{3}{4}$ " end mill, leaving .05 stock, need to be changed to use a 7/8" end mill leaving .062 stock. In operation #30 the drilling needs to be done to a depth of Z-.6 rather than the current Z of -.75.

### <u>Step 26</u>

In the Job Processing dialog, click on the [Operations Manager] button. Select operation #10, then click on the "Details" page. Change the details page to look as follows, then close the operations manager.

Machining Operations Manag	ger			- 🗆 X
Operation List Detai	S Material Tooling	Tool Shape		
Operation # 10 Fixture Offset 0 \$ Feed * IPM ^ IPR 8.00000 Decemining:	Operation Type Milling  Tool Diameter 0.87500 Spindle / RPM C C Forward	Tool Number 1 Stock Allowance 0.06250 Speed Ss 2000 Range #1 •	Offset Number 1 Corner Radius 0.00000 Coolant : Flood	
ROUGH PROFILE .75 E	ND MILL		Set <u>H</u> ome Set <u>G</u> age	

# <u>Step 27</u>

On the Job Processing dialog, double click on the line under operation #30 that reads "Drill Linear". First the linear drilling dialog will be displayed, press [OK] to leave the current settings. Next the Drill Depth dialog will be shown. Click on the box that reads "Absolute – Z- Position to Drill To" and change it from -.75 to -.6, press [OK].

rilling Cycle / Depth Selection	1	×
Drill Cycle Type	Drill Depth	Drill Depth
Standard Drilling 🗨	Rapid Plane (-R-)           0.1           Use AltR- Plane           0.10000	SPECIFYING DRILL DEPTH
Deep Drill Peck Increment	Drill Tip Angle :	
Tapping TPI: 16.0 Pitch : 0.0625	-Z- Top of Part: Specify Depth By: 0.00000 Through Depth to Drill Through Center Drill Information	
Dwell Time	Note: Only Center Drills Large / Small Enough to generate the Hole Diameter Speficied Above Will be Shown	A35 - Z-
Defaults OK	Cancel Help	

### <u>Step 28</u>

In the Job Processing Dialog, press the [Reprocess Job] button. In a few seconds the entire tool path will be regenerated.



## <u>Step 29</u>

At this point, the new tool path is generated. To regenerate a new NC-Code file, simply repeat step #22 (Run the post processor).

# DIMENSIONING

With the machining done, we will probably want to produce a dimensioned drawing. We will add dimensions, a title block, and a side view to the drawing, then make a hard copy plot on the printer.

### <u>STEP 30</u>

While dimensioning we do not want to see the tool path, so we will turn off its layer, and set the a new default layer (#101) for the dimensioning. Start by pressing the  $\leq F9 \geq$  key to display the layer control dialog. First to change the current layer, type 101 in the "Current Layer" box, Click on the "description" column for layer 101 and enter "dimensioning". Finally click on the box displayed in the "ON" column, in the Layer 100 row, until the "X" is not displayed in the box.

Num	ON	Color	Used	Style	Description	<b>▲</b>	
96	X			Solid			All <u>O</u> n
97	X			Solid	T		
98	X			Solid	T		All O <u>f</u> f
99	X			Solid	T		
100				Solid	Machining		Show <u>U</u> sed
101	X			Solid	Dimensioning	·····	L
102	X			Solid	T		
103	X			Solid	T		_ ⊂Current Layer
104	X			Solid	T		
105	X			Solid	T		101 🌻
106	X			Solid	T		
107	X			Solid	T		Line Style
108	X			Solid	T		
				F	T	·	L

At this point the display should look like figure 4-31



Figure 4-31 Display with the tool path layer off

# <u>STEP 31</u>

On a color display, it is generally appealing to have the dimensions displayed in a different color that the part, so press the  $\leq F8 \geq$  key and select a new color.

# <u>STEP 32</u>

First we will dimension the overall width of the part. Select "DRAFTING" from the main menu, then "DIMENSION" then "Horizontal" from the sub menu.

For the **"First Extension Line Origin"** select **"END-OF"** from the sub menu then pick the lower left corner of the part (figure 4-32). Next select **"END-OF"** and pick the lower right corner of the part.

Indicate the approximate text point (See figure 4-33)

The dimension should look like figure 4-34



Figure 4-32 Extension line origin



Figure 4-33 Indicating the approximate text point



Figure 4-34 Dimensioning the overall width

# <u>STEP 33</u>

Now we will dimension the overall height. Select "DIMENSION" then "Vertical" from the sub menu.

For the **"First Extension Line Origin"** select **"END-OF"** from the sub menu then pick the lower right corner of the part (figure 4-35). Next select **"END-OF"** and pick the upper right corner of the part.

Indicate the approximate text point (See figure 4-35)

The dimension should look like figure 4-36



Figure 4-35 Constructing the vertical dimension



Figure 4-36 Dimensioning the overall height



Figure 4-37 Dimensioning the top most line

# <u>STEP 34</u>

Try to dimension the top most line of the part using step 29 and figure 4-37, as examples.

# <u>STEP 35</u>

Now we will add the radius dimensions. Select "**DIMENSION**" then "**RADIUS**" from the sub menu. When asked to "**Select a Arc**", move the pick box over the 1/2" radius and press the left mouse button.

Indicate the approximate text point as shown in figure 4-38

# <u>STEP 36</u>

Dimension the 1 3/8" radius as in step 32.

Indicate the approximate text point as shown in figure 4-39



Figure 4-38 Dimensioning the 1/2" radius



Figure 4-39 Dimensioning the 1 3/8" radius

#### 56 POWERSTATION

### <u>STEP 37</u>

Now let's dimension the center of the 1 3/8" radius and center of the first hole. Select "DIMENSION" then "ORDINATE" from the sub menu.

You will be given the "GET POINT" menu to select a base point for the ordinate dimensions. Select "END-OF" from the sub menu and pick the lower left corner of the part.

The message area of the screen will read:

#### "Point for ordinate Dimension, Right button when done"

Select "END-OF" from the sub menu and pick the bottom of the center mark. Indicate the approximate text point as shown in figure 4-40

Next select "END-OF" from the sub menu and pick the right most point on the arc center mark. Indicate the approximate text point as shown in figure 4-41

Next dimension the first hole by selecting "CENTER" from the sub menu for both the X and Y dimensions.

Press the right mouse button and select <Escape> from the popup menu to end the ordinate dimensioning mode.

The part should look like figure 4-42



Figure 4-40 Dimensioning the X center of the 1 3/8" radius



Figure 4-41 Dimensioning the Y center of the 1 3/8" radius



Figure 4-42 Ordinate dimensions

# <u>STEP 38</u>

Now let's dimension the angle. Select "DIMENSION" then "ANGULAR" from the sub menu.

Select Line #1 (select the line to the right of the 1/2 radius)

Select Line #2 (select the line to the left of the 1/2 radius)

Indicate the approximate text point as shown in fig 4-43



Figure 4-43 Constructing the angular dimension

# <u>STEP 39</u>

Now we will dimension the distance between the holes. Select "**DIMENSION**" then "**HORIZ**" from the sub menu.

For the **"First Extension Line Origin"** select **"CENTER"** from the sub menu, and pick hole #3. For the **"Second Extension Line Origin"** select **"CENTER"** from the sub menu, and pick hole #4. Next indicate the approximate text point as shown in figure 4-44



Figure 4-44 Dimensioning the distance between holes

# <u>STEP 40</u>

Now we will add the text "typical" under the last dimension. Select "TEXT" from the sub menu.

fting - Text		
Text Justification C Left Right C Center Alligned	Text Height :       0.12500         Rotation Angle :       0.0         Slant /Tilt Angle :       0.0	Text   Fonts
Place By Cursor Drag		Text Examples
Enter Text :	Special Symbols [frome] Insert Insert Special Character	Aligned & Justification Aligned & Left Fight, Slanted 20° & Center Slanted -20° &

Next enter the approximate text start and end points as shown in fig 4-45

### 60 POWERSTATION **STEP 41**

Now to add a finished look to our drawing, let's merge in a title block and sheet frame. Select "GEOMETRY" from the main menu, then "BLOCKS" then "MERGE" from the sub menu. A standard WINDOWS file open dialog will appear select the file named "SIZEB.PSB", then press the [OK] button.

When asked "Select the Block Insertion Point", select "COORDS" from the sub menu, and answer:

-X- <u>2.5</u> -Y- <u>-4.5</u>

Scale :	1.00000	
Angle :	0.00000	
Angle :	0.00000	<u></u>



Figure 4-45 Adding text



Figure 4-46 Title block and sheet frame

#### 62 POWERSTATION

#### STEP 41 continued . . .

Notice that we are seeing very little of the block that has been merged. This is because most of it extends beyond the current display. To fit the entire drawing on the display, press the right mouse button anywhere within the main display area, and select "Zoom Extents" from the popup menu. See figure 4-46

### <u>STEP 42</u>

Now let's generate a side view. Select "Geometry", "Shapes", "Box" from the menu.

Entry Method	Coordinates	
C Specify Width & Height	Width: 0.75000	
C Specify Second Corner		L
Enter Coordinates	Heigth: 4.00000	Box Width
	Base -X- : 8 0000	K
Base Point Defines	1	
Corner Point of Box	Base -Y- : 0.00000	Box Height
Center Point of Box	,	Radius/Chamfer Size
		×
Corner Shape	° Chamfor	
	Cirdinici	
Box Corner Radius/Chamfer : 0,1	D	Base Angle
Bass Apple .		×
	D	Base Point of Rotation
		Same as First Point
		(Corner or Center)
		· · · · · · · · · · · · · · · · · · ·

The display should now look like figure 4-47



Figure 4-47 Adding a side view

### <u>STEP 43</u>

Now we will dimension the part thickness using the side view we generated. Select "DRAFTING" from the main menu, then "DIMENSION" then "Horizontal" from the sub menu.

When asked for the "First Extension Line Origin", select "END-OF" from the sub menu, and pick the top left corner of the side view.

When asked for the "Second Extension Line Origin", select "END-OF" from the sub menu, and pick the top right corner of the side view.

Next indicate the approximate text point as shown in figure 4-48

#### <u>STEP 44</u>

Now we will add the drawing name to the title block. This will be easier to do if we "ZOOM IN" on the title block. To zoom, press the right mouse button, anywhere within the main display area, and select "Zoom" from the popup menu. Using the mouse, select any two diagonal corners of the title block.



Figure 4-48 Dimensioning the part thickness

ISSUE	REVISION		DATE	ΒY	СНК′Ъ
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APPR.	TITLE				
DATE					
SYSTEM					
SCALE	SIZE	DRWG	NO,		

Figure 4-49 Zooming in on the title block

# STEP 44 continued . . .

To add the text, select the "DRAFTING" from the main menu, then "TEXT" from the sub menu. You be given the text entry dialog:

CLeft		For Forts
C Diaht	l ext Height : 0.12500	
C Center	Rotation Angle : 0.0	
Alligned	Slant /Tilt Angle : 0.0	
Place By Cursor Drag		Text Examples
Enter Text : LESSON #1	Special Symbols [rone] Insert Insert Special Character	Alighed B Justification Alighed B Justification Fight Bight Bigh

Select the text start point and end point as shown in figure 4-50.

## <u>STEP 45</u>

To return the display to the previous view, press the right mouse button, and select "Zoom previous" from the popup menu.

### <u>STEP 46</u>

Save the drawing as in step 15.

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DRAWN			тки	ΝЛ	
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DATE				. 1	
SYSTEM		22UN	++	Ĺ	
SCALE	SIZE	DRWG NO,			

Figure 4-50 Adding the drawing name

Note: For information on how to permanently change the default title block to have your own company name/logo, please see the "Applications Cookbook".

### <u>STEP 47</u>

Finally let's print the drawing. To print, select "File", "Print" from the menu.

The "display" file generation will take a few seconds. When done, the printing will start.

#### 66 POWERSTATION

# LATHE TUTORIAL



Figure 4-60 The Sample Lathe Drawing

The purpose of the tutorial is to guide you step by step through the creation and machining of a lathe part. The drawing shown in figure 4-60 will be taken from the initial geometry creation, to machining, and finally generate the NC-Code. Each step will be numbered for easy reference.

#### <u>STEP 1</u>

We begin by starting POWERSTATION. To do this: Press [Start] (on the WINDOWS Tool bar), then select "Programs", then "MICROCIMM", then "POWERSTATION"

**Note:** The following steps show one of many "long Hand" methods of developing the part geometry and trimming it. These methods are shown primarily as an example of how to build geometry step by step. This method has application to everything from the simplest to the most complex parts. However: In a case like the sample lathe part we are working on here, POWERSTATION has a much simpler and quicker way of generating the part geometry. After completing the following steps, please take a look at step 10 it shows how to generate the same part geometry using far fewer steps, using the built in "Easy Geometry" command.

### 68 POWERSTATION <u>STEP 2</u>

From the menu, select "Geometry-Line-Polar" The "Get Point" menu will be displayed, select "Origin (X/Y Zero)". Next enter the polar coordinates for the end of the line. Note that the distance is .5 or  $\frac{1}{2}$  of the diameter.

End Point of Line	×
<b>Distance</b> : 0.50000	]
<b>Angle</b> : 90.00000	
OK Cancel <u>H</u>	elp

# <u>STEP 3</u>

From the menu, select "Geometry-Line-Polar" The "Get Point" menu will be displayed, select "End of", and select near the end of the line drawn in step 2.. Next enter the polar coordinates for the end of the line.

End Point of Line	×
Distance :	1.00000
Angle :	180.00000
	Cancel <u>H</u> elp

### <u>STEP 4</u>

From the menu select "Display-Zoom In" to make the geometry a bit easier to see. Next from the menu, select "Geometry-Line-Tangent to One" The system will display "Select a Point or Arc, or <Escape> For Point Menu". The "Get Point" menu will be displayed, select "End of", and select near the end of the line drawn in step 3.



Next you will be asked to enter the angle of the line (150 or -30 are both correct):



The display should look like this:



## <u>STEP 5</u>

From the menu, select "Geometry-Line-Polar" The "Get Point" menu will be displayed, select "Coordinates", and enter the following:

Coordinate Entry ×			
-X- Coordinate -4.75000			
-Y- Coordinate 1.50000			
Enter Point Coordinates			
OK Cancel <u>H</u> elp			

Notice that the -Y- is the diameter divided by 2.

Next enter the Polar coordinates:

End Point of Line	×
Distance :	3.00000
Angle :	0.00000
	Cancel <u>H</u> elp

Select "Display-Zoom Extents" and the display should look like the following:



# <u>STEP 6</u>

Next we will neaten up the geometry by trimming it. From the menu, select "Modify-Trimming-Easy Trim". Clean up the geometry by selecting it at the three points indicated:



# <u>STEP 7</u>

Next we will insert the two fillet radii. From the menu, select "Modify-Trimming-Fillet" You will be asked to select the two lines (P1 & P2 below) and then enter the radius (0.25). Repeat using points P3 and P4.





After Filleting
### <u>STEP 8</u>

Finally we will add the Chamfer on the face. From the menu select "Modify-Trimming-Chamfer". Select the line on the face (P1), and then the 1" diameter (P2).



Next enter the Chamfer distance & angle.

Chamfer Distance & Angle	Х
Distance : 0.12500	
Angle : 45.00000	
OK Cancel <u>H</u> elp	



# <u>STEP 9</u>

To be safe, lets save the file. From the menu, select "File-Save As" and enter "Lesson2"

### <u>STEP 10</u>

The following is an alternate method for building the exact same geometry, using the "Easy Geometry" feature. (This following procedure can be done IN PLACE of steps 2-8 as previously outlined).

Select "Geometry-Easy-Geometry-Lathe Geometry" from the menu.

Enter the following settings into the Easy Geometry Dialog (Note, be sure to move the highlight to a different (any) other field after entering information into the "Entry" column. If you compare these entries to the sample drawing you will see that you can make quick work out of basic turning features.

<sup>E</sup> z Easy Geometry	z Easy Geometry (Lathe Mode)						
Feature	Entry	^	Add Delete				
Diameter -X-	▼ 1.0000						
Chamfer -Dist-	▼ 0.1250		Show Extents				
Shoulder -Z-	▼ -1.5000		<u>H</u> elp File				
Fillet -Radius-	▼ 0.2500		OK Cancel				
Taper -Angle-	▼ 150.0000						
Diameter -X-	▼ 3.0000						
Fillet -Radius-	▼ 0.2500		🔽 Origin @ X0/Y0				
Shoulder -Z-	▼ -4.7500						
[		~					

### 76 POWERSTATION

### <u>STEP 11</u>

No matter how you got here, this is what the geometry should look like at this point.



## MACHINING

POWERSTATION uses a method of operation modeled after the way a machinist thinks. We call this method "OPERATION ORIENTED". Note: The remainder of this section does not apply to POWER-CAD users. The first step is to define one more operations that will take place. The second step is to select the operation that you want to work on, and finally do the machining for that operation.

Note that the operations must be specified in the order that you want the part machined in, but they do not have to be selected & worked on in any specific order. POWERSTATION will automatically generate the operations in the originally defined sequence no matter what order the operations were selected in.

If you make a mistake while doing any machining operations, first try pressing the <F7> (DELETE-LAST) key. If this does not produce the desired results, simply select the "Machining", "Tool Path|Edit", "Delete-Operation" command to delete all tool motion from the current operation, re-select the operation and try again.

Select "**Machining**", "**Operations**" from the menu (or press the operations ICON on the icon bar (located on the left side of the screen). Not sure which is the "operations" icon ? Simply hold the mouse cursor over the icon in about two seconds, a "Hint" will be displayed describing the function of the icon.

Machining Operatio	ns Manager				-	×
<b>b b</b>	✓					
Operation <u>L</u> ist	<u>D</u> etails	Material	Tooling Tool	l <u>S</u> hape		
Current Operation	n # Post	Color Desc	ription			^
		÷				 
		÷				 
		÷				 
		_				× .

Figure 4-89a (The Machining Operations Manager)

📕 Machining Opera	tions Manager				_	×
Operation <u>H</u> elp						
👿 🙀 🕒	<ul> <li>✓</li> </ul>	<u>مار</u>				
Operation <u>L</u> ist	<u>D</u> etails	Material	Tooling Tool	<u>S</u> hape		
Current Operat	ion # Post	Color Desc	ription			^
Yes 10	X	Roug	h Turn			
		- <del> </del>				
						 - 1
						 -
	1					1 M .

*Figure 4-89b (Basic operation information)* 

To enter an operation:

- 1) Click the cursor in the "Op #" column, and type in the number of the operation (10).
- 2) Double click the mouse on the "Color" column, and select the desired color.
- 3) Click the cursor on the "Description" column, and enter a description of the operation
- 4) Double click on the first "Current" column, and the word "Yes" will appear. This indicates that this is now the currently selected operation.
- 5) At this point the dialog should look like figure 4-89b6) Click on the tab that reads "Details". This will switch to the "details" page where you enter the specifics on the currently selected operation (tool number, diameter, etc..). See figure 4-89c

### Lathe Tutorial 79

Machining Operation	s Manager			-	×
<u>8</u> <b>B</b>					
Operation <u>L</u> ist	Details Material Tooling Too	ol <u>S</u> hape			
Operation #	Operation Type Turning	Tool Number	Offset Number		
Fixture Offset	Tool Radius	Stock Allowance	Lathe Comp Mode		
Feed © IPM C IPR 0.01200	Spindle / Speed C RPM © CSS 5 V Forward Rand	d 50 ge #1	Coolant : Flood 🗸		
Description:					
Rough Turn			Set <u>H</u> ome		
			Set <u>G</u> age		

*Figure 4-89c (Operation #10 Details)* 

Machini	ing Operations Help	Manager		>	×
1		✓	۳.	<u>ada</u>	
Operat	tion <u>L</u> ist <u></u>	etails	Mate	rial Tooling Tool <u>S</u> hape	
Current	Operation ‡	Post	Color	Description	^
Yes	10	X		Rough Turn	
	20	×		Finish Turn	
	30	X		0.060 Grooving Tool	
	40	X		Threading	
			, , ,		
					~

# Figure 4-89d (All operations for the tutorial part)

80 POWERS	STATION			
Machining Operation	ns Manager		_	×
🖹 🙀 🕒				
Operation <u>L</u> ist	Details Material Tooling	Tool <u>S</u> hape		
Operation #	Operation Type Turning 🗸	Tool Number	Offset Number	
Fixture Offset	Tool Radius	Stock Allowance	Lathe Comp Mode	
Feed © IPM C IPR 0.01200	Spindle / C RPM © C V Forward	Speed SS 550 Range #1 -	Coolant : Flood 💌	
Description:				
Finish Turn			Set <u>H</u> ome	
			Set <u>G</u> age	
1				

*Figure 4-89e (Operation #20 for the tutorial part)* 

Next define the remaining operations, using the information displayed in figures 4-89 c,e,f & g.

💋 Machining Operations Manage	r			-	$\times$
Operation <u>H</u> elp					
Operation List Details	Material Tooling T	ool <u>S</u> hape			
Operation # 30	Operation Type Turning	Tool Number	Offset Number		
Fixture Offset	Tool Radius	Stock Allowance	Lathe Comp Mode		
Feed FIPM C IPR 0.00500	Spindle / Spe C RPM © CSS	ed 550 nge #1 -	Coolant : Flood 🗸		
0.060 Grooving Tool			Set <u>H</u> ome		
			Set <u>G</u> age		

Figure 4-89f (Operation #30 for the tutorial part)

Lathe	<b>Tutorial</b>	81
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🕖 Machining Operation	ns Manager	– 🗆 X
Operation <u>H</u> elp		
<u>*</u>		
Operation <u>L</u> ist	Details Material Tooling Tool Shape	
Operation #	Operation Type Tool Number Turning V 4	Offset Number 4
Fixture Offset	Tool Radius         Stock Allowance           0.03125         0.00000	Lathe Comp Mode None
Feed • IPM C IPR 0.01200	Spindle / Spe <u>ed</u> © RPM C CSS 678   Forward Range #1 •	Coolant : Flood 🗸
Description:		
Threading		Set Home
		Set <u>G</u> age

*Figure 4-89g (Operation #40 for the tutorial part)* 

### <u>STEP 12</u>

Next, select "Operation-Set Home For All" home position of X6 and Z6.

### <u>STEP 13</u>

At this time it is a good idea to make a print out of the defined operations. To do this select "Operation", "Print" from the menu.

#### <u>STEP 14</u>

Next we need to make operation #10 active. To do this, double click on the "Current" column for operation #10, then exit from the Machining Operations Manager.

### <u>STEP 15</u>

It is a good practice to place your machining on a different layer than the part geometry. While we are dimensioning the part we really do not need to see the tool path, so placing it on its own layer gives us an easy way to "hide" the tool path.

Num	ΟN	Color	Used	Style	Description 🔺	
98	X			Solid		All <u>O</u> n
99	×			Solid	тт	
100	×			Solid	Machining	All Off
101	×			Solid		
102	×			Solid		Show Used
103	×			Solid		
104	×			Solid	T	
105	×			Solid	T	-Current Layer
106	×			Solid	T	
107	×			Solid	тт	100 🌩
108	×			Solid	T	
109	X			Solid	T	Line Style
110	X			Solid		

To change the current layer to 100, press the  $\langle F9 \rangle$  key (or select "Display", "Layer-Control" from the menu), then click on "Current Layer" box and type: <u>100</u>, click the cursor on the description box (The right most column in the layer 100 row), and enter in a description like "Machining". Next press the [OK] button.

### <u>STEP 16</u>

This is a good time to save our work as in step 9.

### <u>STEP 17</u>

With our roughing tool, we will now face off, then rough turn the part. First the facing. Select "Machining" - "Lathe Cycles – Lathe Facing" from the menu. Lets assume a rough bar stock diameter of 3.5 and additional rough stock on the face of 0.25.

e Facing	
✓ Cut Toward Spindle Center Line         Largest Diameter :       3.50000         Smallest Diameter :       0.00000         Stock Allowance :       0.03000         -Z- at Face of Part :       0.00000         -Z- at Face of Stock :       0.25000         Maximum Depth of Cut :       0.12500         Clearance Amount :       0.10000	Lathe Facing -Z- Face -Z- Face of Stock Largest Dia. Max Depth of Cut Smallest Dia.

Ш

#### 84 POWERSTATION

### <u>STEP 18</u>

Now let's start the roughing cycle. Select "Machining", "Lathe Cycles", "Lathe Roughing/Undercutting" from the menu. The single most important rule for turning work is *"IF THE TOOL IS NOT GOING TO TOUCH IT, THEN DO NOT SELECT IT*" (In fact don't even define it).

Mode         Tool Orientation         Roughing Direction           © Roughing         © External (0.D.)         © Turning           © Undercutting         © Internal (I.D.)         © Facing	0.D. Roughing   I.D. Roughing   0.D. Undercuting   I.D. Undercutting   Rough Direction
Depth of Cut Per Side : 0.12500 Stock Allowance in -X- : 0.01500 Stock Allowance in -Z- : 0.00800 Clearance Amount : 0.10000 Bar Stock Diameter : 3.50000	Bar Stock Dia. Rough Boundary
Image: Control of the control of t	

Select the starting line or arc of the boundary to rough (See figure 4-93.



Figure 4-93 Selecting the starting point for roughing

#### Select the line or arc to stop before, <Escape> for none

Press the <Esc> key or press the right mouse button, and select <Escape> from the menu.



### Note: You must select at least two entities (lines or arcs)

At this point the display looks like figure 4-94, and you will be asked to indicate the side of the boundary to offset to. Select a position just to the right of the front face and the center line.



Figure 4-94 Proper chaining



Figure 4-95 Picking the side to offset to



Figure 4-96 Roughing completed

### <u>STEP 19</u>

Next we will bring the tool back home. Select "Machining", "MOVE & CUT" - "GO HOME" from the menu.

### <u>STEP 20</u>

Select operation 20 from the "Machining", "Operations" Dialog.

### <u>STEP 21</u>

On many slower computers, the display of the tool images (small circles) can be very time consuming. If you want to turn them off, select "**DISPLAY**" - "Tool Image" from the menus.

Tool path Disp  Show Tool	ilay Path Onlu	
C Show Tool	Image	
C Show Machined Path		
C Show Full Animated Tool Image		
🔽 Display	Tool Path	

### <u>STEP 22</u>

Finish cutting the O.D.. Select "Machining", "MOVE & CUT" - "CHAIN" from the menus.

### Select the starting Line or Arc (See figure 4-97). Select the line or arc to stop before, <Esc> for none (Press the <Esc> key or click the right mouse button, and select <Escape> from the popup menu)



#### Pick the Side to Offset to

(Move the cursor just to the right of the front face)



Figure 4-97 Selecting the start line for chaining



### <u>STEP 23</u>

Next we will bring the tool back home. Select "Machining", "MOVE & CUT" - "GO HOME" from the menus. The screen should look like this:



# <u>STEP 24</u>

Select operation 30 from the "Machining", "Operations" Dialog.

#### 90 POWERSTATION

### <u>STEP 25</u>

Now we will generate the groove. Select "Machining", "Lathe Cycles", "Lathe Grooving/Partoff" from the menu. The dialog will be displayed:

Lathe Grooving		×
Groove Type © Straight Groove on the Diameter © Straight Groove on the Face © Contour Groove on the Diameter © Contour Groove on the Face © Part Off Cycle	Contour Groove Options © Rough & Finish C Rough Only © Finish Only V Rough Right to Left Internal Groove	Straight on Dia Sraight on Face Contour on Dia Contour on Face Part Off
Right Side of Groove in -Z- : Left Side of Groove in -Z- : Major Diameter Groove Depth: Groove Corner Radius/Chamfer:	-1.16000         -1.25000         1.00000         0.09000         0.00500         □ Chamfer	Groove Depth Left Side -Z- Right Side -Z- Major Dia. Radius/Chamfer
Tool Width : Tool Corner Radius : Clearance Amount : Dwell Time at Groove Bottom : Defaults	0.06000 0.00300 0.10000 0.00000 ОК Сапсе! <u>Н</u> еір	

Note: From the way in which the groove is dimensioned, it is probably easiest to enter the - Z- sides of the groove in the following manner.

```
Right Side of Groove in -Z- : -1.5+.09
Left Side of Groove in -Z- : -1.5
```

These expressions can always be replaced by the exact decimal number if you know it. If you do not know it, why not let the computer do the math for you ?

#### <u>STEP 26</u>

Next we will bring the tool back home. Select "MOVE & CUT" - "GO HOME" from the menu.



Figure 4-99 The grooving complete

# <u>STEP 27</u>

Select operation 40 from the "Machining", "Operations" Dialog.

### <u>STEP 28</u>

Now we will add the thread. Select "Machining", "Lathe Cycles", "Lathe Threading" from the menu.

athe Threading		×
Athe Threading         Thread Pitch         Threads Per Inch:         Thread Pitch:         0.08333             Depth - Infeed / Outfeed         //Depth are Far Sider         Total Depth:         0.05112         First Pass Depth:         0.01400         Clearance:         0.1         In-Feed Angle:         30.0	Thread Diameter         Image: Diameter	Threading Tool End -Z- Start -Z- Diameter
Out-Feed Angle :     0.0       Pull Out, In-Threading Mode       Defaults         Calc 75% Depth	Canned Cycle Options : [0-Standard "Long Hand" G32/33 Type	→   ← Pitch Total Depth

Notes on threading: After entering in the threads per inch, press the "Calc 75% Depth" button. This will take the current thread pitch and calculate the proper depth for a 75% of a standard 60 degree thread. Also, when entering the "Left Side (End)", again try entering the location as an expression "-1.25+.045" in order to thread into middle of the groove.

### <u>STEP 29</u>

Next we will bring the tool back home. Select "MOVE & CUT" - "GO HOME" from the menu.



Figure 4-100 The Threading Complete

### <u>STEP 30</u>

Once again, lets save the file as in step 9. After that, lets check the machining status. Select "Machining", "Machining Status" from the main menu. When done viewing the status, press the [OK] Button.

### <u>STEP 31</u>

The final step in machining is to run the post processor. Select "Machining", "Post (Generate NC-Code)" from the menu (or select the Post icon).

From the list of machines in the upper left hand corner of the dialog box, move the highlight to "DLATHE.MCH".

Next we will set the name of the NC-tape file to be created. Press the button labeled [Set NC-Code File Name]. Enter "LESSON2" as the "File Name", and press [OK].

Press the [Go] button. The post processing now begins, and may take a few minutes. When done, press [Close] to leave the post processor.

ost Processor		×	
Machine Name	NC-Code File Name :		
Dedm.mch	lessonz. I AP		
DLATHE.MCH	Set NC-Code File Name		
DMILL.MCH			
Line Number : 413			
Error Messages:			
NC-Code			
N1320 Z.1		^	
N1330 TO M05			
N1340 M30			
8		~	
<		>	

#### 94 POWERSTATION

Now that the tutorial part is finished, lets see how the new "Job Processing" feature can help with making changes.

### <u>Step 32</u>

Select "Machining-Job Processing". The main display will change in order to accommodate the Job Processing Dialog.



### <u>Step 33</u>

In the Job Processing Dialog, in the "Tree area" (Upper left) one at a time, click on the Plus "+" signs. The dialog will now look as follows:



#### 96 POWERSTATION

### <u>Step 34</u>

For this example, we will make three common changes and show how quickly we can regenerate the tool path and the NC-Code. For this example, lets assume the following changes:

- 1. The tool nose radius on the roughing tool was changed to 1/16
- 2. The a rough bar stock diameter turned out to be 3.75 not the expected 3.5
- 3. The additional rough stock on the face turned out to be .375 not the expected 0.25.
- 4. The depth of cut for the roughing need to be increased to 0.2

#### <u>Step 35</u>

In the Job Processing dialog, click on the [Operations Manager] button. Select operation #10, then click on the "Details" page. Change the details page to look as follows, then close the operations manager.

💋 Machining Operation	is Manager	-	$\times$
Operation <u>H</u> elp			
Operation <u>L</u> ist	Details Material Tooling Tool Shape		
Operation #	Operation Type     Tool Number     Offset Number       Turning     1     1		
Fixture Offset	Tool Radius     Stock Allowance     Lathe Comp Mode       0.06250     0.02000     OD Turning •		
Feed IPM C IPR 0.01200 Description:	Spindle / Speed Coolant : C RPM © CSS 550 Flood V V Forward Range #1 V		
Rough Turn	Set Home Set <u>G</u> age		
1			

# <u>Step 36</u>

On the Job Processing dialog, double click on the line under operation #10 that reads "Lathe Facing". Change the settings to the following to reflect the new bar stock size.

Lathe Facing	Х
🔽 Cut Toward Spindle Center Line	Lathe Facing
Largest Diameter : 3.75000	
Smallest Diameter : 0.00000	
Stock Allowance : 0.03000	-Z- Face -> -Z- Face of Stock
-Z- at Face of Part : 0.00000	Largest Dia.
-Z- at Face of Stock : 0.37500	Max Depth of Cut
Maximum Depth of Cut : 0.12500	Smallest Dia.
Clearance Amount : 0.10000	
Defaults OK Cancel Help	

### <u>Step 37</u>

On the Job Processing dialog, double click on the line under operation #10 that reads "Lathe Roughing". Change the settings to the following to reflect the new bar stock size and the new depth of cut.

Lathe Roughing Operations			×
Mode To	ol Orientation Roug External (O.D.) © Tu Internal (I.D.) C Fa	ghing Direction urning acing	0.D. Roughing ] I.D. Roughing ] 0.D. Undercuting ] I.D. Undercutting ] Rough Direction ]
Depth of Cut Per Side : Stock Allowance in -X- : Stock Allowance in -Z- : Clearance Amount : Bar Stock Diameter : Stock on Bar Face : V Cutting V Cutting Select S V Use Car	0.20000 0.01500 0.00800 0.10000 3.50000 0.00000 From Right to Left Part Origin at X0/Z0 tock Boundary med Cycle If Available	Help	Bar Stock Dia.

### <u>Step 38</u>

In the Job Processing Dialog, press the [Reprocess Job] button. In a few seconds the entire tool path will be regenerated.

### <u>Step 39</u>

At this point, the new tool path is generated. To regenerate a new NC-Code file, simply repeat step #41 (Run the post processor).



# INDEX

А  $ALL \cdot 13$ angles · 19 ANGULAR · 58 ARC · 25 В  $BLOCK\cdot 20$  $box \cdot 22$ BY-LAYER · 13 BY-TYPE · 13 С CAD/CAM  $\cdot$  5, 20 calculator  $\cdot$  18 CENTER · 42, 56, 58 CHAIN · 13, 38 colors  $\cdot$  50 D default directory · 11 Degrees · 18 Delete last · 27 diameters  $\cdot$  19 DIMENSION · 50, 52, 54, 58 dimensioning  $\cdot$  49 display file;, select .i.FILE · 65 DRAFTING · 50 drilling  $\cdot$  42, 43 DXF · 8 E END-OF · 50, 56 expression · 18 Extension Line  $\cdot$  50, 58 F FILLET  $\cdot$  29 function keys · 15 G geometry · 21, 67 GEOMETRY  $\cdot$  22, 62 GET DISTANCE · 17 GET POINT · 16, 42, 56 Η HORIZ  $\cdot$  50, 58 Hot keys · 15 Ι  $\text{ICON} \cdot 11$ install · 11 L Laserjet · 8  $LAST \cdot 13$ layer · 37, 82 layers · 20 LINE · 26, 28

100 POWERSTATION Μ machining  $\cdot$  21, 67 MACHINING · 38, 44, 93 main menu  $\cdot$  22  $\text{MODIFY} \cdot 14$ 0 operation  $\cdot$  15 OPERATION  $\cdot$  41  $ORDINATE\cdot 56$ outside  $\cdot$  14 Р  $\text{PAN}\cdot 23$ plot · 49, 65  $\text{PLOT} \cdot 65$ plotted · 21  $\text{POINT}\cdot 16$ POLAR  $\cdot$  17 Post-Processed · 67 printer · 49 prompt  $\cdot$  22 R Radians  $\cdot$  18 RADIUS · 54 REMOVE · 13 Repeats the last command  $\cdot$  15 S save  $\cdot$  33  $\text{scale}\cdot 20$ side view  $\cdot$  49 SINGLE  $\cdot$  13, 14  $\text{STATUS}\cdot93$ Т TANTO 1  $\cdot$  26  $\text{THRU}\cdot 43$ TWO-PT · 17 V  $\text{VERT}\cdot 52$ W window  $\cdot$  14 WINDOW  $\cdot$  13 Ζ Z-Level · 14  $\text{ZOOM} \cdot 20$