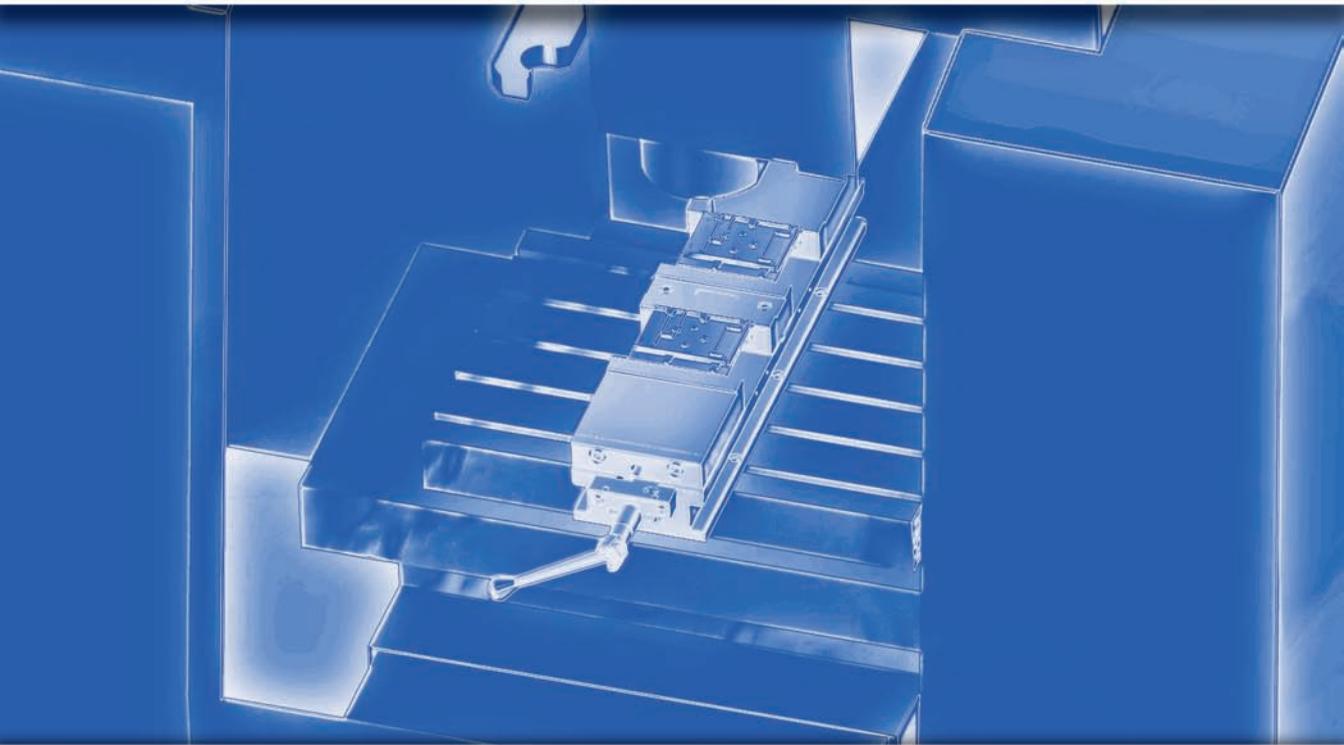


Mastercam X⁴

Handbook Volume 1



Mill Level 1 & Solids

Mastercam Handbook Volume 1 for Mastercam X4

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Software: Mastercam X4 Mill & Solids

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This document complies with Mastercam-X4 as of June 2009. Requires Mastercam Mill Level 1. Requires Solids for Chapter 5

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Introduction

Congratulations on your purchase of the Mastercam Handbook Volume 1. This book was developed and tested over several years as part of a course to teach machinists how to program CNC machine tools with Mastercam. It assumes you have a working knowledge of machining and CNC's, including tooling, work-holding, and common G&M codes. No prior experience with computers or CAD/CAM is necessary, but helpful.

The Handbook can be used as the primary resource for a Mastercam class, as a self-study guide, or a shop reference. Mastercam help functions and information on the student CD supplement the material in this book.

A unique feature of this book is the comprehensive, top-down approach it takes to learning. Concepts and essential knowledge are included along with practical applications. This approach means you not only learn how to use Mastercam, but why things are work as they do.

By understanding exactly what you are doing and why each step of the way, you learn to recognize the best approach to problems. Not only will you be better prepared to work effectively, you will gain a solid foundation of knowledge that will help you continue to learn and adapt as technology changes.

Specific recommendations about how to best use Mastercam are included. There are often many ways to accomplish any task. However, you will learn faster and understand the overall picture of what you are doing if you first master fundamentals and standard practices.

In any case where information in this book conflicts with your machine manuals or the methods used at your company or school, ignore the suggestions in this book and use the information in the manuals and established procedures at your facility.

Never operate a CNC Machine without having read and understood the operator and programmer manual, and having received safety and operator training by a qualified person on that machine.



Warning

Contents

This book is divided into ten chapters. Each covers a specific knowledge area. Following is a breakdown of the chapters and what you will find in each:

Chapter 1: Computer Essentials covers the computer knowledge and skills you'll need to operate a CAD/CAM system.

Chapter 2: Mastercam Workspace shows how to use, navigate, and customize the Mastercam workspace.

Chapter 3: CAD Drawing shows how the Mastercam user interfaces works, how to draw basic geometry such as lines, arcs, and points, and how to import/export data between different CAD/CAM systems.

Chapter 4: Advanced CAD Drawing shows how to move, copy, rotate, mirror, and scale geometry, and how draw basic 3D wireframe geometry.

Chapter 5: Solid Modeling introduces how to draw in a 3D Workspace, create basic solid models, and how to organize and manage your drawings.

Chapter 6: Drill Toolpaths shows how to use hole-making functions including drilling, peck drilling, and tapping.

Chapter 7: Contour Toolpaths shows how to create 2D, 3D, Ramp and Remachining contour toolpaths.

Chapter 8: Pocket Toolpaths teaches basic skills to remove excess material.

Chapter 9: 2d High Speed and FBM teaches parameters which are necessary for setting up these toolpaths.

Chapter 9: Rotary Toolpaths teaches how to create 4th axis indexing and axis substitution toolpaths.

Chapter 10: Change Recognition shows how to identify, isolate, and update changes between part revisions.

Appendices: A-D as shown on the Contents page. App E & F are in PDF format on the student CD only.

Icons are used to alert, inform, and enhance your learning experience. The following icons are found in the margins of the book:

Icons

Remember reminds you of important information that will help you work safely and productively.



Remember

Tips are suggestions from experienced CAD/CAM users that will guide your learning and use of Mastercam.



Tip

Step by Step are detailed instructions on how to use a specific function or perform a task.



Step by Step

On The CD alerts you that a file exists on the CD included with this manual that may be necessary for accomplishing a task.



On The CD

Try It assigns a task you should be able to successfully complete before proceeding further.



Try It

Warning is used to emphasize situations that can cause damage to machines, property, bodily injury or death. Machining can be dangerous. Take these warnings seriously and do not proceed unless you are certain your methods and setup are completely safe.



Warning

In Depth are notes of interest that deepen your understanding and knowledge of a topic.



In Depth

Power User denotes tips that are likely only applicable to the highest level users of Mastercam.



Power User

Conventions Key words and Mastercam menu items are shown in **bold** the first time they are used.

Columns on the outside edges of each page and note pages at the end of each chapter provide ample space for taking notes.

Useful tips, recommended settings, best practices, and detailed instruction on the most important features are included when possible.

3 CAD Drawing

In this chapter, you will learn to draw wireframe geometry. Upon completion of this chapter, you should be able to do the following:

Objectives

- Define elements of the Cartesian coordinate system.
- Understand the difference between absolute and incremental coordinates.
- Define the four quadrants and the sign of points lying within each.
- Correctly determine the Datum on a part print.
- Create lines, arcs, points, rectangles and other geometric shapes.
- View the part from different perspectives
- Use trim, chamfer and fillet functions.
- Dimension a shop drawing.

A computer cannot think. Nor can it scan a drawing and automatically create a usable NC program. The only thing a computer can do is what you, or the person who wrote the software, commands it to do. In this respect, the computer is just like any other tool. It helps you do your job better, and it can make good parts or bad, depending on how you use it.

Introduction

While the computer cannot think, its ability to process huge volumes of information, quickly and without error, is unmatched.

Every bit of information needed to draw and machine your part is stored in a large database that Mastercam manages for you. What you see on the computer screen is a picture of that database.

You work with the picture, not the lists of numbers that make it possible. Behind the scenes, Mastercam responds to your every input, updating the database and changing the picture to reflect every change immediately.

This way of working with a computer is **Interactive**. You instruct the computer to do something, and it does it. You see the results of your actions and decide to undo, change it, or move on to another task. Instead of acting as a human calculator trying to visualize what the numbers mean, you work with pictures that change on your command.

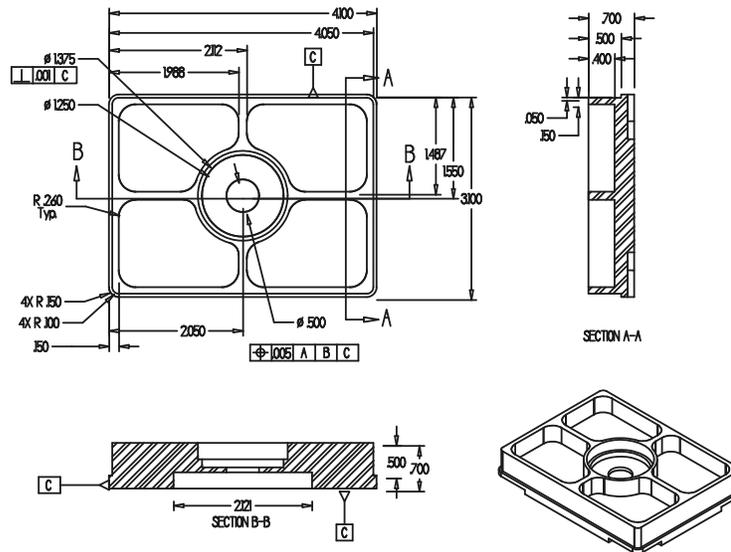
Since humans are visually oriented, this way of working is far more efficient than writing CNC programs by hand. When you do something, you see the results immediately.

Errors are easier to spot. Changes are faster and easier to make. Once you are confident that the machining processes are exactly what you want, the software does the tedious work of writing the CNC program.

With Mastercam, you seldom, if ever, need to use an electronic calculator. Geometry problems are solved using Mastercam's many geometry creations, transformation, and editing tools — not trigonometric calculations.

There is an old saying about computers, "Garbage In, Garbage Out". This means the computer will perform well if you instruct it properly and poorly if you do not.

However, assuming you have done your job well, and your software is setup properly, Mastercam does an excellent job writing CNC programs, even longest and most complex ones, quickly and without a flaw.



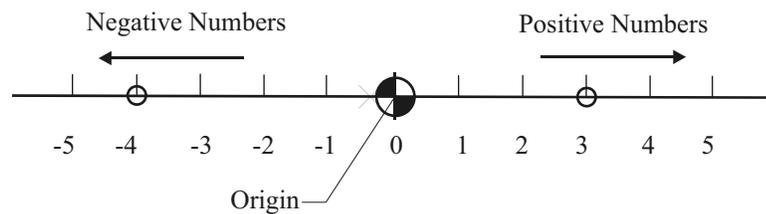
Before learning to draw in Mastercam, review some basic concepts and terms used in CAD/CAM technology:

- Cartesian coordinate system
- Datum
- Planes
- Fundamentals of CAD geometry

Cartesian Coordinate System

For Mastercam to display a part, you must define its exact shape, size and location. Do this by drawing lines, arcs, points, and other geometric entities that precisely describe the part. These geometric entities exist in a Cartesian coordinate system.

A Cartesian coordinate system consists of two or three number lines.



A number line is a line divided into equal segments. Some point on the line is designated as zero. This point is called the Origin. Numbers to one side of the origin are positive. Those on the other side are negative.

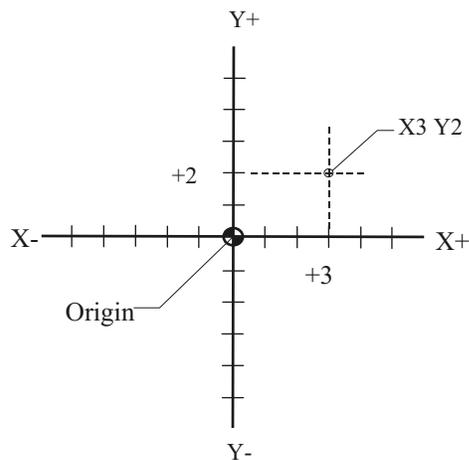
Any point on the line is precisely located given its value and sign. In the example, the coordinate “3” lays three units to the right of the origin point. The coordinate “-4” lays four units to the left of the origin.

It is common practice to drop the sign for positive numbers. Thus +3 is written or entered in the computer as 3. However, negative numbers must include the negative sign “-“. For example, the number -3 must include the “-“ sign.



Remember

A two dimensional Cartesian coordinate system consists of two number lines set at a 90-degree angle to each other. One line is horizontal (left to right) and is labeled the X-axis. The other is vertical (up and down) and labeled the Y-axis. The point where the axes cross is the **Origin**.



Any point in this space, called a **Plane**, or **Construction Plane**, are precisely defined given its axes label, sign, and value. For example, the point “X3,Y2” is located by counting, from the Origin, three units along the X+ axis, then up two units parallel to the Y+ axis.



Tip

Cartesian coordinates may be written two different ways. One uses the axis label, sign and value.

For example: **X3,Y2**

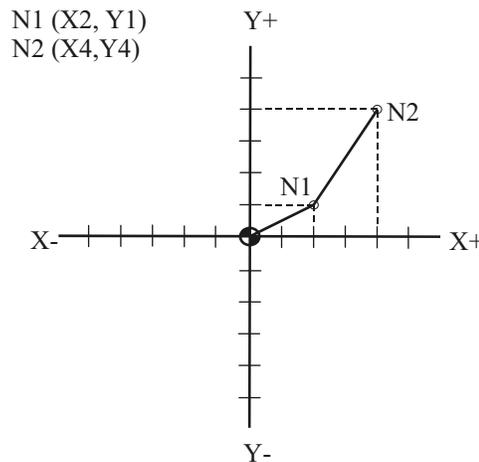
The other writes coordinates as an **Ordered Pair**. Numbers are written in a specific order (X,Y) separated by commas.

For example: **3,2**

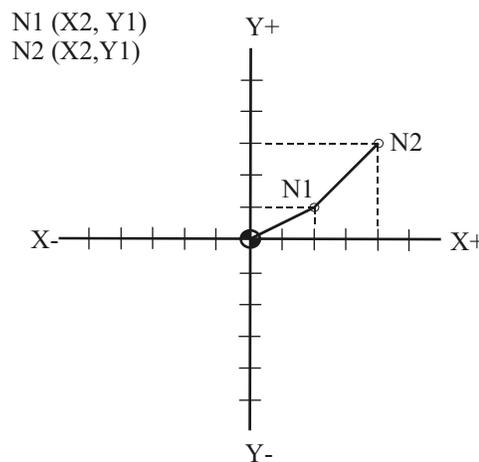
Positions within the Cartesian coordinate system may be described using **Absolute, Incremental or Polar** coordinates.

**Rectangular
Coordinates**

Absolute coordinates are always in reference to the Origin, regardless of the previous position. Starting at the Origin, the following diagram shows a move to N1 and then to N2, written in absolute coordinates.

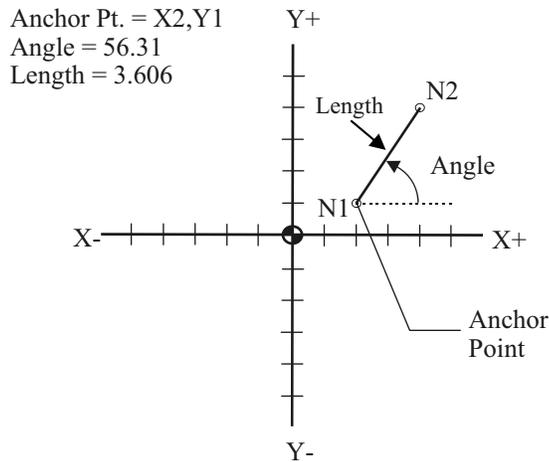


Incremental coordinates (sometimes called Delta or Rectangular coordinates) are always in reference to the current position. For example, starting at the Origin, the following diagram shows a move to N1 and then to N2, written in incremental coordinates.

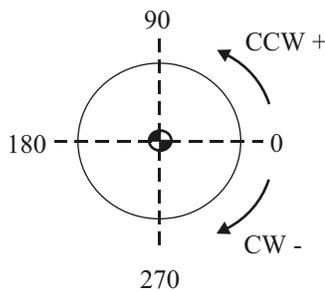


Polar Coordinates

Polar Coordinates are always in reference to a position (called the Anchor Point), a Distance, and Angle. Starting at the position (X2, Y1), the following diagram shows a move to N2, written in polar coordinates.



Angles are measured in degrees from the 3:00 position.

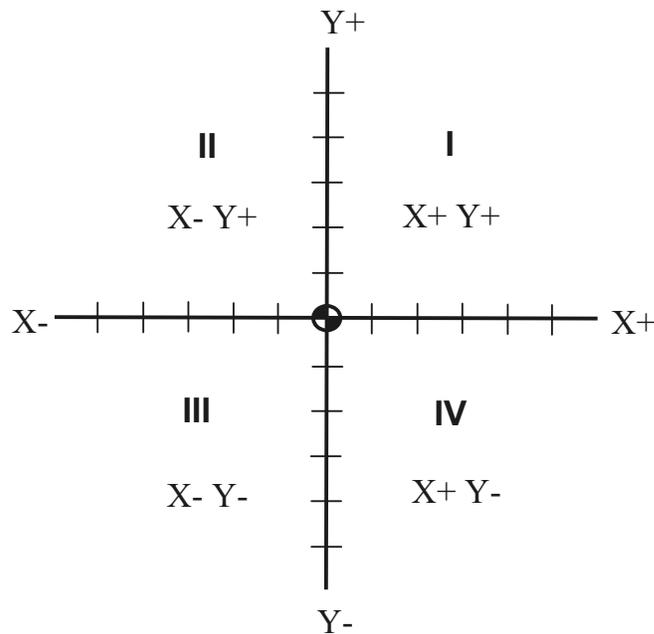


Term	Definition
Angle	CCW angles are positive. CW angles are negative. For example, the angle (315) is the same as (-45).
Anchor Point	Reference position for the polar coordinates.
Degree	1/360 th of a full circle.
Minute	1/60 th of a degree.
Second	1/60 th of a minute. Angles can be expressed in degrees, minutes and seconds, which is abbreviated, DMS.

A Plane can be divided along its axes into four quadrants, starting in the upper-right corner and moving counterclockwise, labelled: I, II, III, IV.

Quadrants

It's important to know which quadrant the part is in because the sign of the coordinates change based on the quadrant. For example, all points in quadrant (I), have positive X and Y values. Points falling in quadrant (II) have negative X and positive Y values, and so on.



Turn to pages 3-59 and 3-60 at the end of this chapter and complete

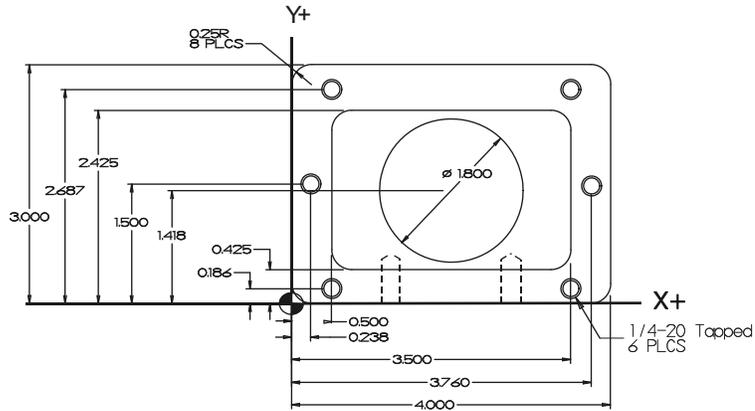
- **Exercise 3-1, Cartesian Coordinate System**
- **Exercise 3-2, Incremental Positioning.**



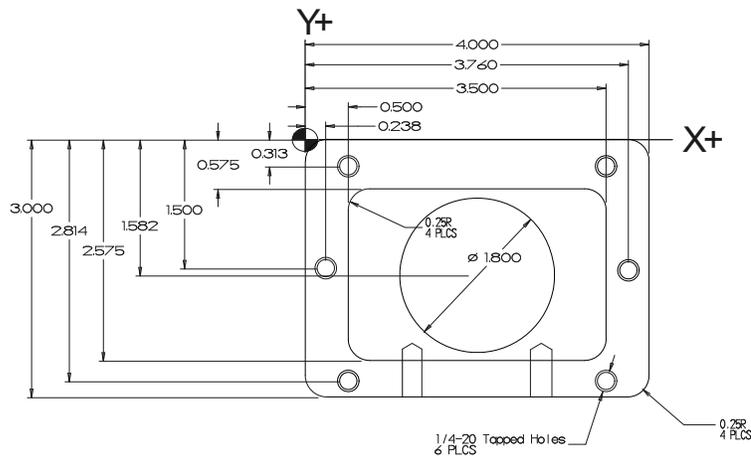
Try It

Datum

The Origin point on a drawing is called the **Datum**. The drawing below shows the datum in the lower-left corner, locating the part in the first quadrant.



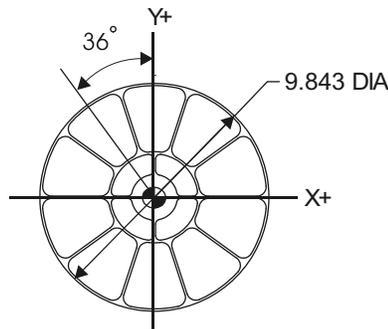
The following drawing shows the same part with the datum in the upper-left corner, locating the part in the fourth quadrant.



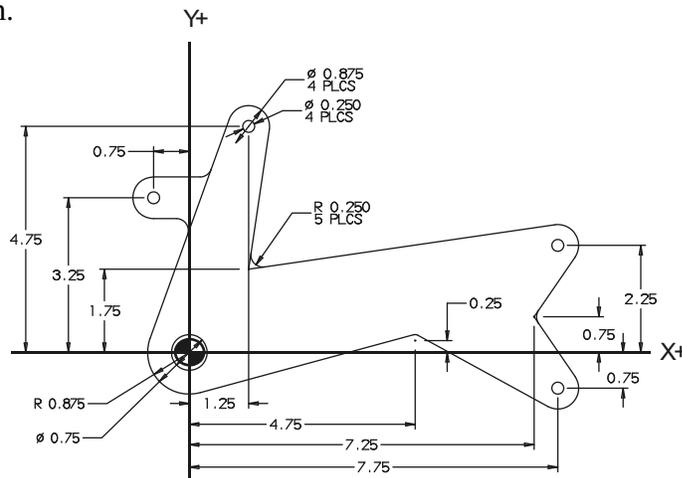
Remember

Even though part prints do not show dimensions as negative numbers, you must input negative values when entering coordinates when appropriate. For example, the hole in the upper left corner in the drawing above is at the coordinate: X.5 Y-.313

Drawings can span more than one quadrant. For example, it is common to place the Datum at the center of round parts.



Since most parts get installed into an assembly, the Datum acts to ensure critical dimensions are held for proper fit and function. In the example below, the critical dimensions are between hole centers in reference to the .75 diameter hole. Thus, the engineer selected the center of this hole as the Datum.



Attention to the datum is essential to part quality. Usually the same datum used to dimension the part is also used for machining.

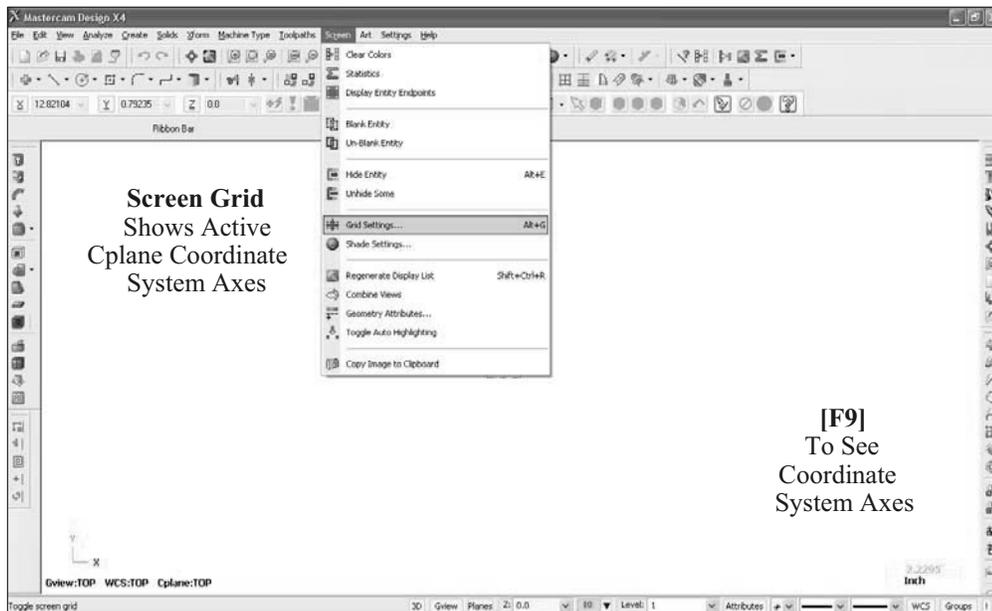


Remember

**Mastercam
Coordinate
Systems**

The Mastercam Coordinate System (**MCS**) comprises the total graphic space that you can work in. It extends, for all practical purposes, infinitely in all directions. Its position and orientation never changes.

Within this coordinate system, any number of **Planes**, called Construction Planes, can be defined. A Plane is a coordinate system that can be located and oriented anywhere within the coordinate system. Planes make drawing easier and are required to define certain 2D entities.



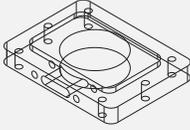
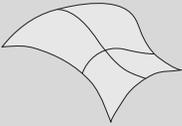
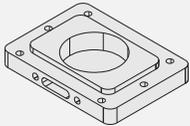
Examples in this chapter use a pre-defined Plane, **Top**. Select the Top Plane by clicking on Plane on the status bar and picking Top from the list.



View the coordinate system axes by selecting **F9** or **Tools, Configuration, Screen, Display part information**. Screen Grid shows the position and orientation of the active Cplane. If active, the Viewport XYZ axes indicator shows the orientation in reference to the active view.

Parts are drawn, or modeled, using geometry. There are three types of geometry used by CAD/CAM software:

- Wireframe
- Surfaces
- Solids

Geometry Type	Description
<p style="text-align: center;">Wireframe</p> 	<ul style="list-style-type: none"> • Wireframe geometry consists of curves (lines, arcs, points and splines) and points. • Wireframe geometry includes information only about the edges of a part. • Wireframe models cannot be shaded. • Wireframe geometry is adequate to model and machine most prismatic or “2-1/2D” parts –where all contours exist in flat planes.
<p style="text-align: center;">Surfaces</p> 	<ul style="list-style-type: none"> • A surface can be thought of as an infinitely thin shell stretched over a wireframe. • Surface geometry includes information about the faces and edges of a part. • There are many types of surfaces; each suited to model a specific type of shape. • Surfaces are used to model complex, freeform (organic) shapes common in the automotive, aircraft, mold, and consumer goods industries. • Surface modeling is covered in the Mastercam Handbook, Volume 2.
<p style="text-align: center;">Solids</p> 	<ul style="list-style-type: none"> • Solids contain information about the edges, faces, and interior of the part. • Most mechanical parts are now designed using Solid Modeling software, like Mastercam Solids, SolidWorks, SolidEdge, ProEngineer, and others. • Solids are able to model many parts, but some highly sculpted shapes, like car bodies, may still require surfaces. • All Solids start with profiles of wireframe geometry. • Solids are covered in Chapter 5, Solid Modeling.

Wireframe Geometry

Wireframe geometry includes the following basic entity types.

Entity	Definition
 Point	A point occupies a single set of coordinates in space. It has no length, depth or width; it is infinitely small.
 Line	A line is an entity defined by any two points in space, called endpoints. Lines have length, but no width or depth; they are infinitely thin.
 Arc	An arc is an entity that is equidistant from a point in space, called a center point. Arcs are “2D” entities, meaning that they must reside on a Cplane to be defined.
 Spline	A Spline is a curve that travels, usually smoothly, through a set of points, called Control Points. There are two types of splines; 2D and 3D. 2D splines are flat entities that must reside on a plane.
 Drafting	Drafting entities include notes, text, leader lines, witness lines, hatch, used to annotate a drawing. Drafting text and notes are stored as a special entity type called a font, which allows lettering to be stored in an efficient format.

Wireframe geometry includes other geometry types, such as a helix, ellipse, and rectangle; but these are modeled using one of the basic types described above. For example, an ellipse is modeled using a spline and a rectangle is modeled using four individual lines.

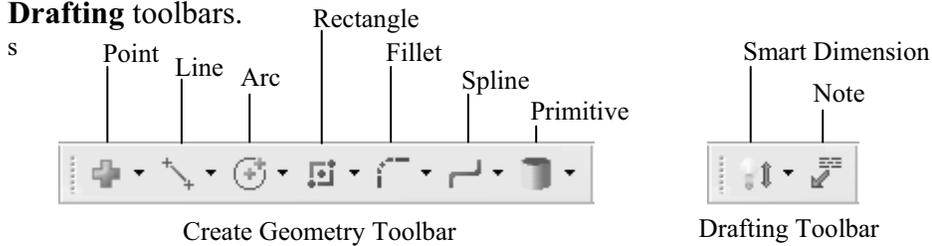
This chapter deals with how to create basic wireframe geometry types listed in the table above. Once you understand these, it will be easy for you to create other types.

Wireframe geometry functions are selectable from the **Create** Menu. The following diagram shows the most commonly used drop down and fly out menus to create basic wireframe geometry.

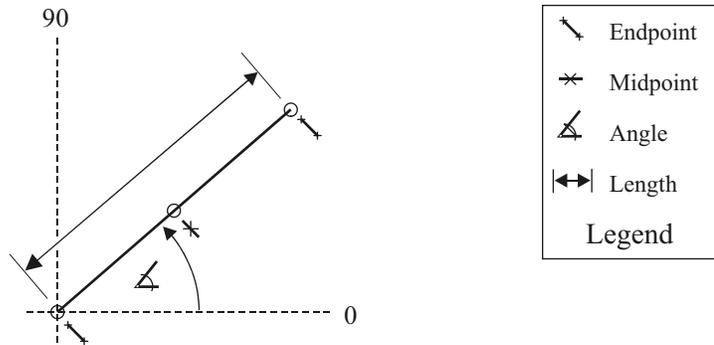
Wireframe Geometry Options



These items can also be selected using the **Create Geometry** and **Drafting** toolbars.



Lines A line is a geometric entity connecting any two points in space. A line can start and end anywhere in the Mastercam Coordinate System.



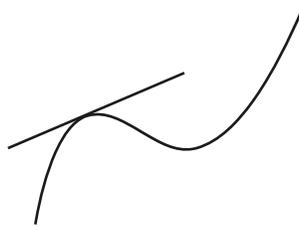
Term	Definition
2D Length	Length of the line in reference to the active view.
3D Length	Full length of the line, regardless of the view. If the line lies in the same plane that it is being viewed, the 2D and 3D lengths are the same.
Angle	The angle of a line is measured from the 3:00 position. Counterclockwise (CCW) angles are positive. Clockwise (CW) angles are negative.
Bisect	A line that splits two other lines equally.
End Point	The coordinates of the either end of a line.
Horizontal	A line along or parallel to the X-axis.
Mid Point	Point equidistant from the end points.
Multi-Line	A series of lines that are connected.
Parallel	A line offset an equal distance from another line.
Perpendicular	A line 90 degrees to another line or arc. Sometimes referred to as a normal line.
Polar Line	A line defined by its start point, length and angle.
Start Point	Lines have a direction. The Start Point is the x,y,z coordinates of the first endpoint.
Tangent	A line that intersects an arc or spline at one point only.
Vertical	A line along or parallel to the Y-axis.

Tangent lines touch an arc or Spline at one point only in the local area.

Tangent



Tangent to Arc

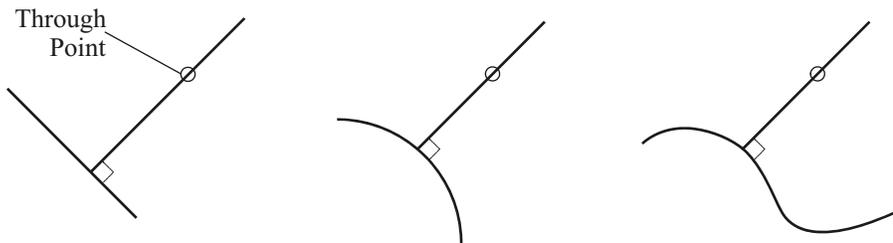


Tangent to Spline

Perpendicular lines pierce a line or curve at a 90 degrees angle all around. In other words, a perpendicular line is a tangent line rotated 90 degrees.

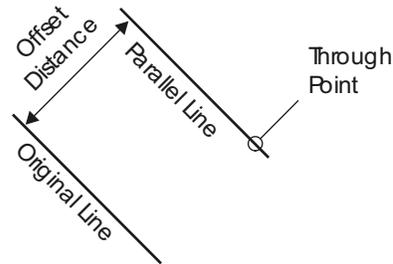
Perpendicular

This type of line is also called a **Normal line** when referring to arcs, splines, or surfaces. Mastercam can create a perpendicular line passing through some point on the curve or a point in space.

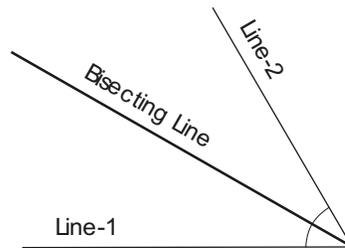


Perpendicular to Line, Arc or Spline

Parallel **Parallel** lines are lines that lie in the same plane but never intersect; regardless of how far they are extended. Mastercam can define a parallel line given an offset distance from an existing line or a through point.



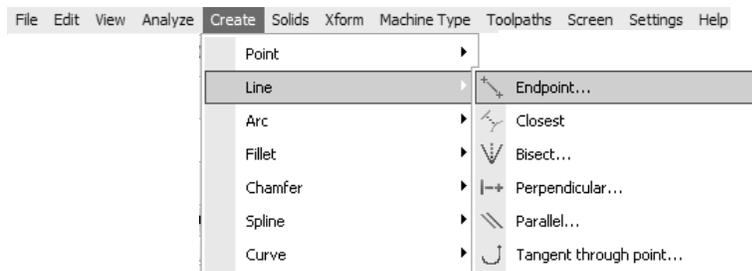
Bisecting **Bisecting** lines split the angle between two existing lines equally. Mastercam shows multiple solutions and prompts to select the one you want.



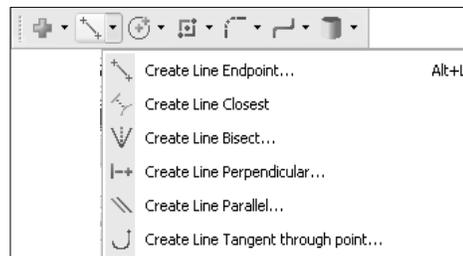
Mathematically, a line has length but no width; it is infinitely thin. When viewed directly along its axis, a line disappears. A line is sometimes referred to as a “straight curve”. A line is a 3D entity; it does not have to lie in a 2D construction plane to exist.

Create lines by selecting **Create, Line** from the Menu.

**Create
Line**



The line options are also available on the **Sketcher** toolbar

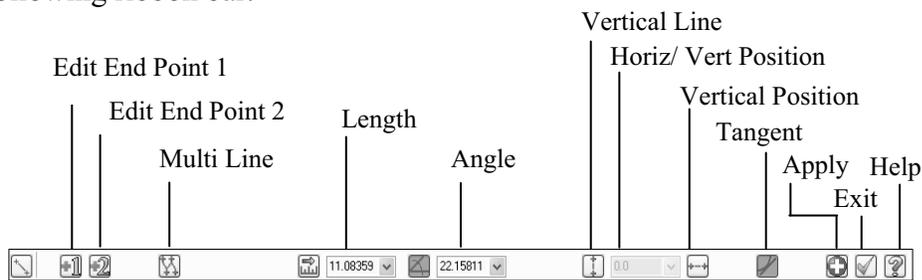


Option	Definition
Create line endpoint	Create a line given its endpoints, length, angle or tangent point.
Create line closest	Create a line representing the shortest distance between two entities.
Create line bisect	Create a bisecting line; a line that splits the angle between two lines equally.
Create line perpendicular	Create a line perpendicular to a line, arc, or spline.
Create line parallel	Create a line parallel to an existing line.

Line Ribbon Bars

The line ribbon bars control values and relations of lines. Line parameters can be changed until the **Apply** or **Exit** buttons are selected to complete its creation. Until then, the entity is said to be “live” and is cyan color. When fully defined, the line changes to the default drawing color.

The ribbon bars change depending on the type of line selected. The **Create line endpoint** option is the most common selection and uses the following ribbon bar.

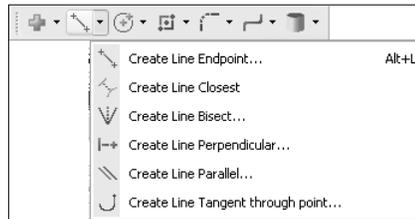


Term	Definition
Create Line Endpoint	Ribbon bar identifier.
Edit Start Point	Change value of start point.
Edit End Point	Change value of end point.
Multi-Line	Create a string of lines.
Line Length	Enter/display length of line.
Line Angle	Enter/display angle of line.
Vertical Line	Draw a vertical line.
Horizontal Line	Draw a horizontal line.
Horizontal/ Vertical Position	X-value of a vertical line, Y-value of a horizontal line. When one of these is active, the horiz/vert position value allows setting the X or Y position of the line.
Tangent	Specify line to be tangent to arc or spline. When this option is active, the line will be tangent to the arc if no other geometric feature, such as an endpoint or quadrant, is selected.
Apply	Create the line but keep ribbon bar open.
Exit	Create line but leave the line create option. Same as selecting the [ESC] key.

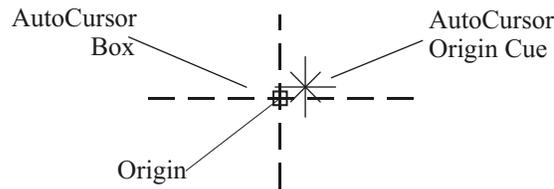
Follow the steps below to create a line given its start point, length, and angle. This exercise is easier if you make the **Screen Grid** visible.

Creating Lines

Step 1: Select the **Create line endpoint** function from the main menu or the Sketcher tool bar.

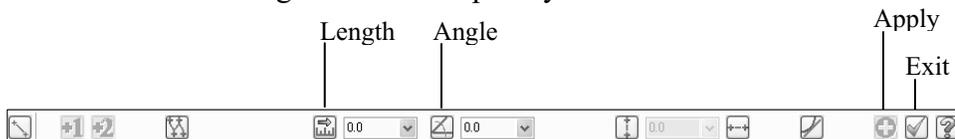


Step 2: Move the mouse near the coordinate system origin until the cursor changes from an arrow to a box, the Origin cue displays, and the small box snaps to the Origin. Click once on the left mouse button.



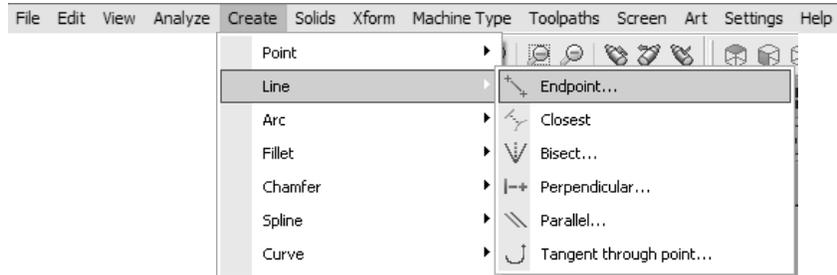
Step 3: Drag the mouse to the right so that the line snaps horizontal and the **Angle** field locks to 0-degrees. Then click once on the left mouse button. A Cyan colored line displays, indicating the line is a “live” entity. Altering parameters on the ribbon bar can still change live entities.

Step 4: Enter **L4** and press **Enter** on the keyboard to set the line length. Finally, click **Apply** to finish the line and remain in this function, or **Exit** to finish the line and leave the function. Notice that the line changes to the active color indicating it is now completely defined.



**More
Line
Options**

Other functions on the line drop down menu are shown below:

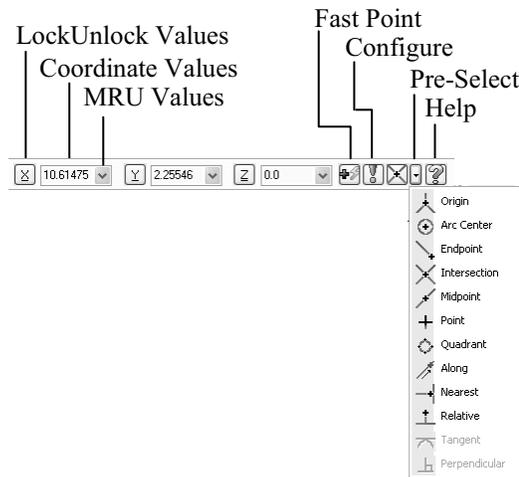


Item	Definition
Create line endpoint	Create a line given its start and/or endpoint, angle, tangent to curve, or other geometric information.
Create line closest	Create a line that is the shortest distance between two entities.
Create line bisect	Create a line that bisects two other lines. When multiple solutions are shown, pick the one you want to keep.
Create line perpendicular	Create a line perpendicular to another line, arc or spline.
Create line parallel	Create a line parallel and at a specified distance from another line.

You should be able to create lines using any of these options by simply following the function prompts. If you have problems, use the on-line help [Alt-H].

During geometry creation, AutoCursor automatically finds and locks (snaps) to geometry features. For example, as the cursor approaches the endpoint of an existing line, the cursor jumps and locks onto it.

An AutoCursor cue appears near the cursor. This cue changes to show the type of geometric feature is located. Click to accept this feature location, or move the cursor to find another feature in the area.

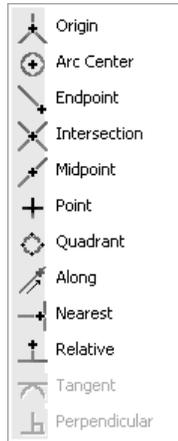


Term	Definition
Lock/Unlock Value	Click the axis label to lock entry in field so it does not change as the cursor moves. Selecting Shift on the keyboard with the axis label does the same.
Coordinate Value	Field entries for X,Y,Z coordinate values.
MRU Values	Click to display the most recently entered values.
Fast Point	This works the same as pressing the spacebar. Enter coordinates as ordered pairs. See the Tip on page 2-3 for coordinate entry rules.
Configure AutoCursor	Configures mouse to select or not select one or more AutoCursor items.
Pre-Select AutoCursor	Click a feature on this drop down list to force AutoCursor to only see that feature for the next mouse selection only.
Help	Help on how to use AutoCursor.

AutoCursor Cues



Mastercam uses the following cues to show which geometric feature the mouse has locked onto:



Term	Definition
Origin	Active coordinate system origin.
Arc Center	Center of an arc or full circle.
Endpoint	The end of a line, arc, or spline
Intersection	Where two wireframe entities cross each other.
Midpoint	Midpoint of a line, arc or spline.
Point	A geometric point entity.
Quadrant	For a full arc, the 0, 90, 180, and 270 degree positions.
Along	Distance along an entity.
Nearest	Closest distance between two entities.
Relative	Relative to a reference point.
Tangent	Tangent to an arc or 2D spline.
Perpendicular	Perpendicular to a line or arc.

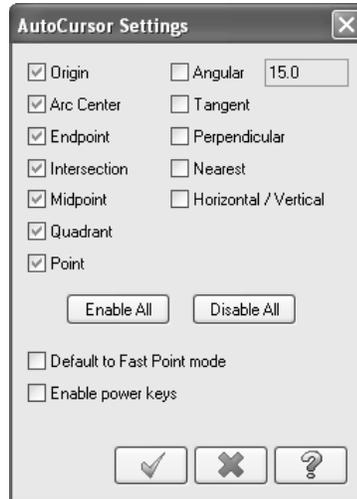


Tip

Click an item on the AutoCursor pre select drop down menu to force Mastercam to find only that item for the next mouse pick. Afterwards, the AutoCursor reverts to the settings in Configure, AutoCursor.

Use **AutoCursor configure** to disable or enable the geometric features AutoCursor will sense.

Configure AutoCursor Dialog



Item	Definition
Enable All	Activates all AutoCursor selections.
Disable all	Disables all AutoCursor selections.
Default to Fast Point mode	Set system to default to FastPoint data entry so coordinates are input manually.
Enable Power Keys	Allows using Power Keys to override AutoCursor settings, for example, E for Endpoint, C for Arc Center.

The Sketch mouse mode is almost never used for entering coordinate data. Instead, use the numeric keyboard or the mouse AutoCursor.

Mastercam stores coordinates to sixteen places. The mouse, and your hand, are not nearly precise enough to sketch coordinates accurately at normal screen extents.



Tip

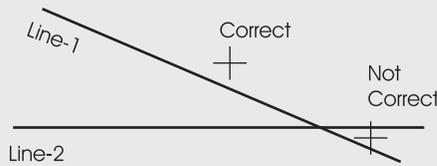
It is common for beginners to move the mouse directly over the endpoint of an entity or intersection point of two entities when using geometry pre-selection. This is not always the best choice, especially in crowded areas of the drawing.

To select Line-1 in the illustration below, it is better to click away from the endpoint to keep from inadvertently selecting Line-2 .



Tip

Use **AutoCursor** to reduce the chance of accidentally choosing the wrong entity.



Entering Numeric Data

Observe the following rules when entering numeric data in Mastercam using the computer keyboard or numeric pad:

Rule	Right	Wrong
Decimal points are not required for integer values.	1	1.
Trailing zeros are not required.	1.3	1.3000
Fractional Values are allowed.	3/8 or .375	-
Positive numbers do not require the plus sign.	1.5	+1.5
Negative numbers require the “-“ sign preceding the number.	Y-1.5	-Y1.5
Enter equations using standard algebraic rules.	(5+.25)/3	-

Undo/Redo

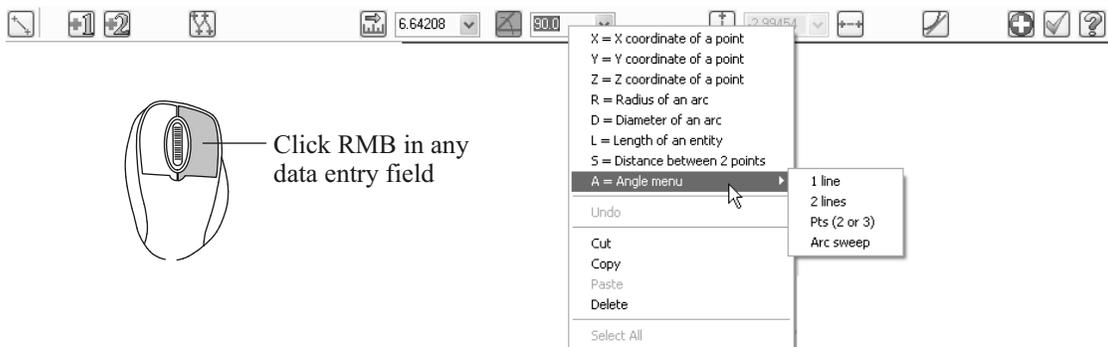


As you work, Mastercam keeps track of your actions. If you make a mistake, click on the Undo icon (left arrow) to undo the action. Continue to click Undo to undo the next previous action. The Redo (right arrow) function reverses Undo actions.

Right click in any data field in Mastercam to retrieve numeric data from existing geometry. For example, you can retrieve the length of a line, the diameter of an arc, or the angle between two lines.

**Retrieving
Numeric
Data**

The illustration below shows a right click with the cursor in the angle field on the line toolbar. This opens a drop down menu. Select the geometry feature to query, and then follow the function prompts.



You can continue to change values on the ribbon bar until the **Apply** option, **[ESC]** keyboard button, or select any other Mastercam function.

The most efficient way to proceed is to simply select another geometry creation option. There is no need to select Apply each time to create the entity.



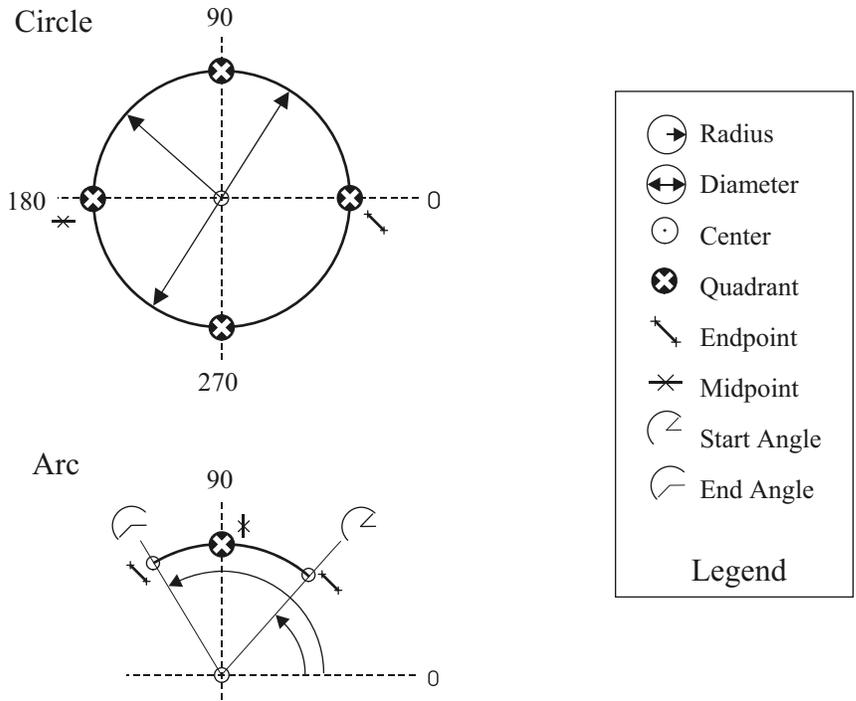
Please complete the following exercises at the end of this chapter before proceeding further:

- **Exercise 3-3, L-Block**
- **Exercise 3-4, Polar Lines**



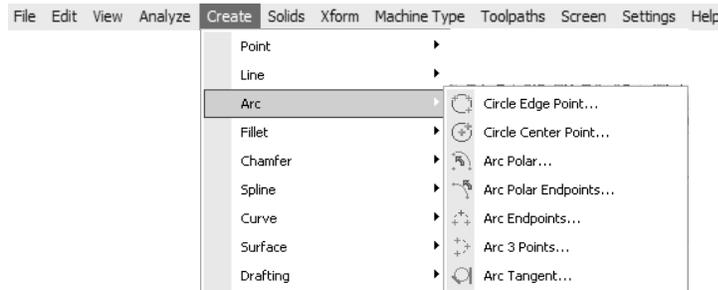
Arcs

An arc is a curve equidistant from a given point, called the center point. Arcs have a start and end angle, measured from the zero degree (3:00) position. A circle is a full 360-degree arc.



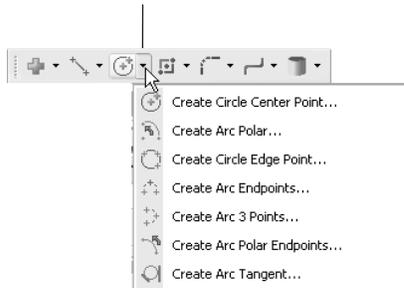
Term	Definition
Center Point	X,Y,Z coordinates of the center of the arc.
Quadrant Point	A point on the arc at the 0, 90, 180 or 270 degree angle.
Circumference	Distance measured around the perimeter of a full arc.
Length	The distance around the arc from the start to the end point.
Mid Point	The point equidistant from the arc start and end points.
Start/End Point	Mastercam places the start/end point at the 3:00 position.
Sweep	Total included angle of the arc.
Radius	Distance from center to edge of arc.
Diameter	Measurement across the widest part of the arc.
Start/End Angle	The start or end angle measured from the 3:00 position.
Pi	Arc circumference/radius. Approximately 3.141593

Create arcs by selecting **Create, Arc** from the menu:



The arc options menu is also available from the Arc option on the **Sketcher** toolbar.

1. Click On Arrow Next to Arc Icon

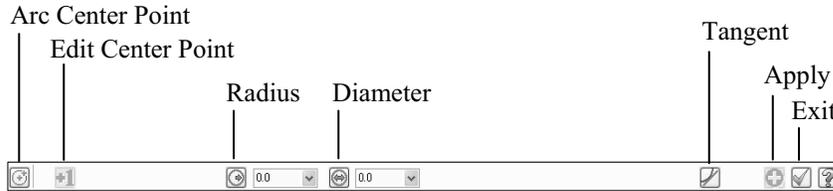


2. Click on arc type desired

Function	Definition
Create circle center point	Create full circle given its center point and radius, diameter, edge, or tangent point.
Create arc polar	Create arc given its start and end angles.
Create circle edge point	Create full circle given its center and edge point.
Create arc endpoints	Create arc given two end points.
Create arc 3 points	Create arc given three thru points.
Create arc polar endpoints	Create arc given start/end points and angles.
Create arc tangent	Create an arc tangent by a variety of means.

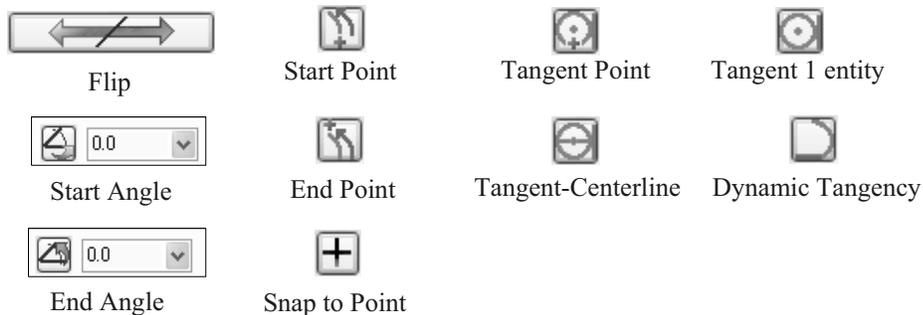
Arc Ribbon Bars

The **Arc** toolbar controls values and relations of arc entities as they are being created. The arc ribbon bar changes depending on the type of arc selected. **Create circle center point** function is the most common selection. It uses the following ribbon bar:



Item	Definition
Arc Center Point	Function Icon: identifies this as the Create circle center point ribbon bar.
Edit Center Point	Edit center point while arc is live.
Radius	Radius input. Click icon or press space bar and R to jump to this field and lock the value.
Diameter	Input diameter. Click icon or press space bar and D to jump to this field and lock the value.
Tangent	Prompts to select a line or arc for the circle to be tangent to.
Apply	Create arc but keep ribbon bar open.
Exit	Complete the arc.

Depending on the arc option selected, the following options may appear on other arc ribbon bars.



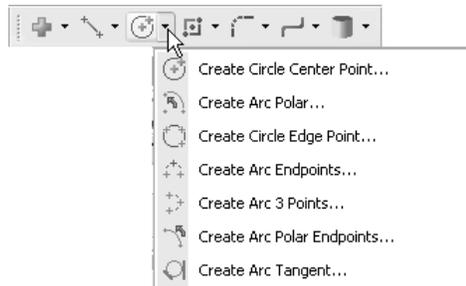
Follow the steps below to create a circle with center point at (X1,Y1) and diameter of 0.5 inches.

Creating Arcs



Step by Step

Step 1: Select the **Create arc circle center point** function from the main menu or the Sketcher toolbar.



Step 2: Enter **D.5** on the keyboard and press the Enter key to set the arc diameter to 0.5 inches. If creating several arcs with the same diameter, press **Shift D** instead to lock the diameter.



Step 3: Press the spacebar on the computer keyboard to open Fast Point and enter 1,1. Press the Enter key. You can now drag the 0.5 diameter arc around the graphic screen.

Step 4: Complete the arc using any of the following options:

- Click on the green check mark on the **Create arc circle center point** ribbon bar shown in Step 2: above.
- Press the Esc key on the keyboard.
- Press the space bar again and enter the coordinates for another arc.

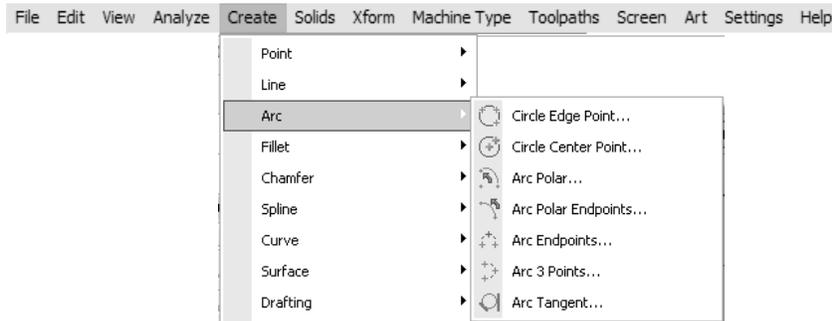
Arcs are 2D entities—they must lie in flat construction plane (view) to exist. You will learn more about Planes in Chapter 4.



In Depth

**More
Arc
Options**

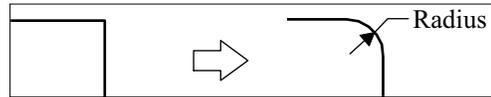
Other arc options on the arc drop down menu are shown below:



Item	Definition
Create arc circle edge point	Creates an arc given its center point and a point the arc intersects. Selecting the tangent item allows the arc to be defined tangent to three entities.
Create arc polar	Creates an arc given its center, start angle, and end angle.
Create arc endpoints	Create an arc given endpoints and radius or diameter.
Create arc 3 points	Create a partial arc given three points on the arc.

You should be able to create arcs using any of these options by simply following the function prompts. If you have problems, use the on-line help [Alt-H].

A **Fillet** (rhymes with skillet) is an arc between two entities. The most commonly used fillet is the normal fillet with trimming, which blends and trims two entities as shown:



The fillet ribbon bar controls the radius, fillet style, and trim choice.

Fillet Ribbon Bar



Mastercam has four trim styles:

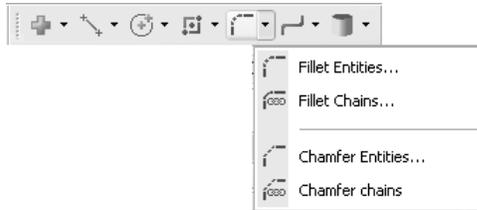
Style	Definition
 Normal	Fillet entities.
 Inverse	Fillet used to relieve corners, often used on the inside corner of sheet metal parts for bending.
 Circle	Create a full circle.
 Clearance	Fillet where center of arc is at the intersection point of the two entities; used to relieve corners so a square object can rest squarely against the two inner walls.

Creating Fillets



Follow the steps below to create a normal fillet between two lines.

Step 1: Select the **Create, Fillet entities** function from the main menu, or the Sketcher toolbar.



Step 2: Set the fillet **Radius**, **Trim**, and **Style** on the Fillet ribbon bar.



Step 3: Move the mouse near the intersection of the entities until a preview of the fillet appears on the screen.

Step 4: Click once on the left mouse button and then **Accept** to create the fillet.

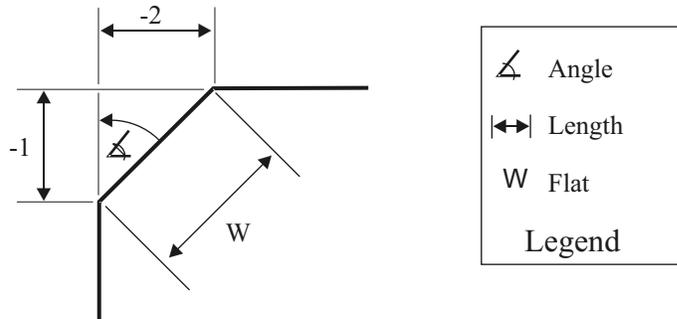


Tip

Place the cursor so a fillet preview appears but neither entity changes color. If you click while one of the fillet entities colors change, then you must select the other entity to complete the fillet. This mode of operation is used to force the system to select a specified fillet when automatic preview does not present the fillet solution needed.

Chamfer creates a beveled edge between two entities. The Chamfer function works similar to the Fillet. Create a chamfer by selecting the **Create, Chamfer** drop down menu or by choosing the Chamfer icon from the **Sketcher** toolbar.

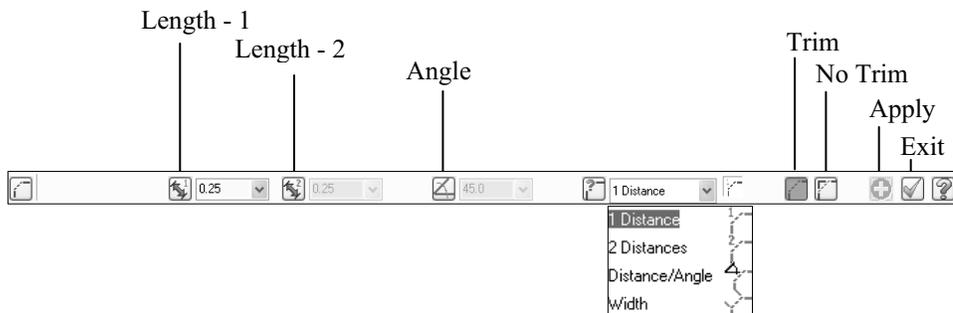
Chamfer



Item	Definition
Angle	Angle the chamfer line forms with either of the lines.
Length -1	Distance across one side of the chamfer.
Length -2	Distance across the other side of the chamfer.
Width	Distance across the chamfer flat.

The chamfer ribbon bar works similar to Fillet and provides several ways to define the chamfer. Select the chamfer type and fill in any open fields. Function prompts will pop up in the graphic area to help you select entities to create the chamfer.

Chamfer
Ribbon
Bar



Point



A **Point** is a geometric entity that occupies a single X,Y,Z coordinate in the workspace. Points can serve many purposes; like locating a drill Mastercam displays points using one of eight point styles. The type used is usually a matter of personal preference. Some point styles are easier to see than others in different conditions. For example, you might find the circle or box style easier to see—especially when working on complex 3D parts.

Set the point style by clicking on the arrow to the right of the Point Style drop down menu on the Status Bar.

	Style	Definition
	3D Point	Default point display option. Can be difficult to see if at the end of a line entity.
	Filled Square	Small filled square. Stays the same size regardless of zoom.
	Cross	Two crossed lines.
	X-Box	Hairline crossed lines with solid box in center.
	Circle	Small circle that stays the same size regardless of zoom. Useful for 3D drawings.
	Open Square	Small square that stays the same size regardless of zoom. Useful for 3D drawings.
	Thread Wire	For Wire EDM programming. Specifies wire thread position.
	Cut Wire	For Wire EDM programming. Specifies wire cut position.

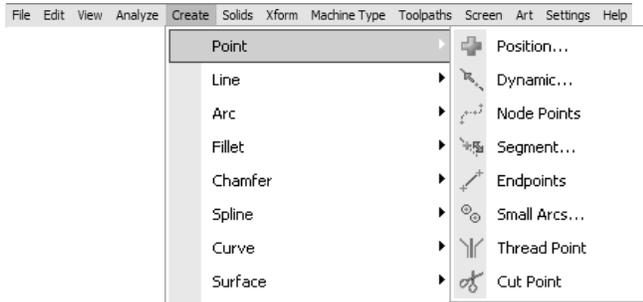


In Depth

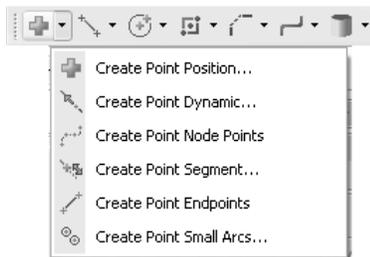
A point has width, depth or length. It is infinitely small.

Create points by selecting **Create, Point** from the Menu.

Point Options



Point options are also available from the **Point** option on the **Sketcher** toolbar.



Item	Definition
Position	A point defined by its X,Y, and Z coordinate.
Dynamic	Place a point along an entity by dragging the cursor along the entity and clicking.
Node Points	Node points are used to create splines. Create a geometric point entity at each Node point for the spline.
Point Segment	Creates points based on a specified number or distance between the points along an entity.
Endpoints	Create a point at the end of each entity. This is very useful for finding chaining problems.
Small Arcs	Create a point at the center of small arcs.
Create Thread Point	Used by Mastercam Wire EDM to indicate a wire thread location.
Create Cut Point	Used by Mastercam Wire EDM to indicate a wire cut location.

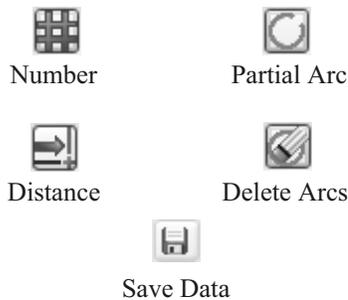
**Point
Ribbon
Bars**

The **Create point position** ribbon bar below allows editing of the point coordinates while the point is live.



Item	Definition
Create Point Position	Function Icon: Identifies this as the Create point position ribbon bar.
Edit Position	Allow editing of the point coordinates while the entity is still live.
Exit	Create the point and exit this ribbon bar.

Depending on the point option selected, the following options may appear on other point ribbon bars.



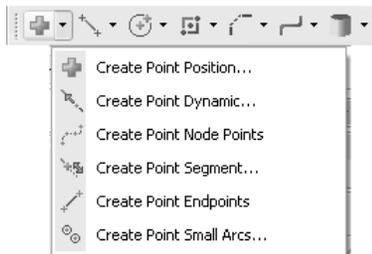
Follow the steps below to create a point given its X,Y coordinates.

Creating Points



Step by Step

Step 1: Select **Create, Point, Create point position** from the main menu or from the **Sketcher** tool bar.



Step 2: Press the spacebar on the computer keyboard to open Fast Point and enter 1,1. Press the Enter key. This places a point at the coordinate X1, Y1.



Step 3: Complete the point by selecting the green checkmark on the Point ribbon bar.



More Point Functions

You should be able to create points using any of these options by simply following the function prompts. If you have problems, use Mastercam help (Alt-H).

Create point dynamic, node pts and segment are mostly used for complex 3D modeling and machining, and are covered in more depth in the **Mastercam Handbook Volume 2**.

Trim/Break

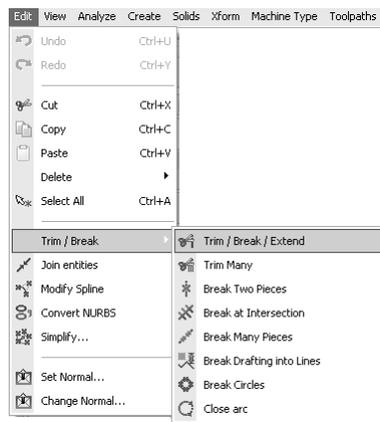
In Mastercam, entities are trimmed or extended using the same **Trim/Break** function.



Trim/Break Options



To trim or extend entities, select the **Edit, Trim/break** drop down menu:

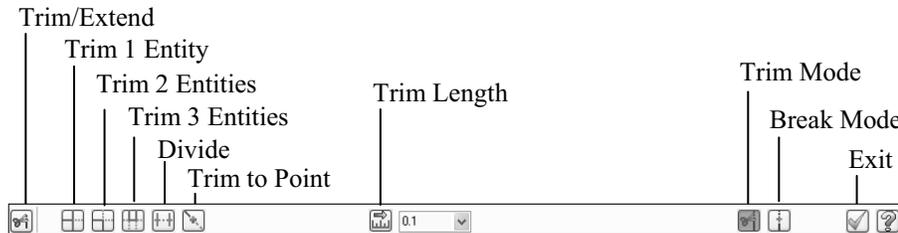


Trim/Break options are also selectable from the **Trim/Break** toolbar.



Select the trim-extend function from the **Trim-Extend** ribbon bar. Like other ribbon bars, this changes depending on the trim/extend function selected.

Trim/Break Ribbon Bar



Item	Definition
Trim 1 Entity	Trim or extend one entity to another.
Trim 2 Entity	Trim or extend two entities to each other.
Trim 3 Entity	Trim or extend three entities to each other.
Divide	Remove a segment of an entity that crosses two other entities. Just click on the part of the entity that you want to go away. Note: Depending on the number of entities, it may take several seconds for this function to find the intersecting geometry and trim.
Trim to Point	Trim an entity back to a point.
Trim Length	Reduce or extends an entity by a specified length.
Trim Many	Trim many entities to one entity. You will be prompted to select all of the entities to trim and then for the entity they are trimmed back to. Finally, you will indicate which side to keep by left clicking with the mouse.

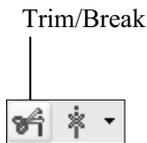
Trimming



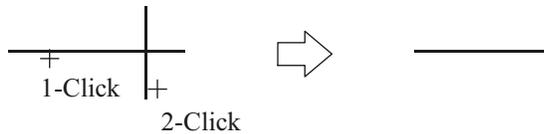
Step by Step

Use the Trim/Break/Extend function to trim one or two entities to each other. To trim one entity to another, do the following:

Step 1: Select **Edit, Trim/Break, Trim/Break/Extend** from the main menu or the **Trim/Break** toolbar.



Step 2: Click once on the first entity (1) and then once on the second entity (2) as shown in the illustration below.



Simple trim-extend operations do not require pre-selecting the Trim 1 or Trim 2 options. Simply select the Trim/Extend function and click once on the first and once on second entity. To trim both, click once on the first entity and twice on the second entity.

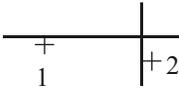
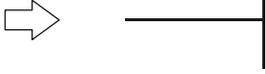
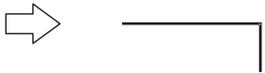
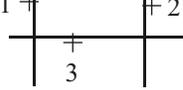
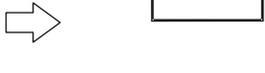
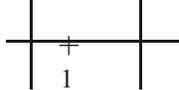
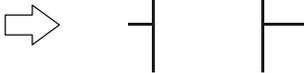
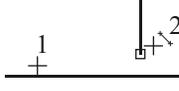
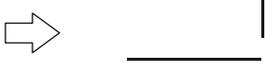
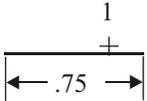
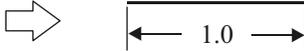


Tip



Always pick the side of the entity to keep. The illustrations below show examples of each trim function. The diagram on the left shows the pick order and approximate click location. The diagram on the right shows the result of the operation.

**More
Trim/Break
Functions**

Icon	Function	Pick Order	Result
	Trim 1 Entity		
	Trim 2 Entity		
	Trim 3 Entity		
	Divide		
	Trim to Point		
	Length <input type="text" value=".25"/>		

Please complete the following exercises at the end of this chapter before proceeding further:

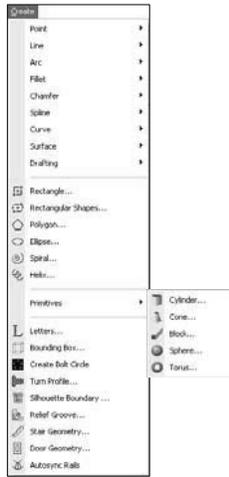
- **Exercise 3-5, Gusset Plate**
- **Exercise 3-6, Control Horn**
- **Exercise 3-7: Reinforcing Plate**
- **Exercise 3-8: Bell Crank**



Try It

More Wireframe Geometry Options

Mastercam has many wireframe geometry functions to make your job easier. For example, the **Create rectangular shapes** function creates rectangles, obrounds, and D-holes. You could create these objects using line and arc commands. However, the rectangle shapes function makes creating these shapes easier by defining them with settings and parameters.



Item	Definition
Rectangle	Creates any of the following objects: Rectangle, Obround, Single D-hole, Double D-hole
Polygon	Creates a multi-sided shape given the number of sides and included radius.
Ellipse	Creates an ellipse given the major and minor radius.
Spiral	Creates a constant or variable pitch spiral.
Helix	Creates a constant or variable pitch helix.
Primitives	Creates basic surface and solid shapes based on parameters. These functions are covered in this Mastercam Handbook Volume 2.
Letters	Creates letters made of lines, arcs and Splines. These letters can be machined.
Bounding Box	Creates a cube with lines that enclosed the objects selected.

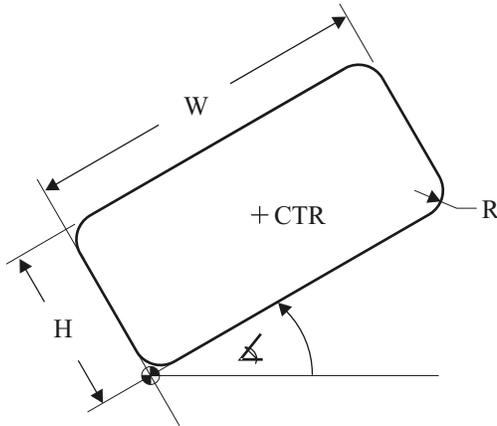
Use the **Rectangle** function to create any of the following shapes.

Rectangles



- Rectangle
- Rectangle with corner radii
- Obround
- Single D-hole
- Double D-Hole

A simple rectangle consists of four lines connected and perpendicular to each other. In Mastercam, these lines are not associated. You can move or delete any individual line without affecting the others.

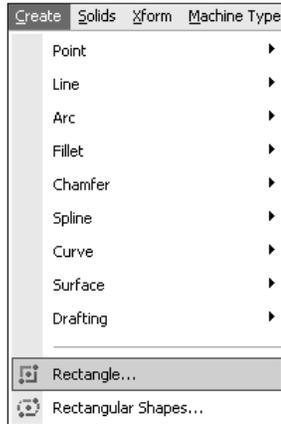


W	Width
H	Height
R	Corner Radius
CTR	Center Point
	Rotation Angle
	Anchor Point
Legend	

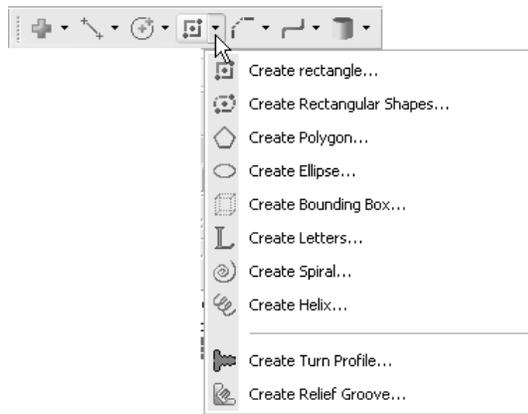
Item	Definition
Width	Distance across one side of the rectangle.
Height	Distance across the other side of the rectangle.
Corner Radius	Radius of corners.
Rotation Angle	Rotation from horizontal 3:00 position.
Anchor Point	X,Y placement point of the rectangle.
Center Point	Point at center of rectangle.

Rectangle Options

Create a rectangle by selecting the **Create, Create Rectangle** drop down menu:

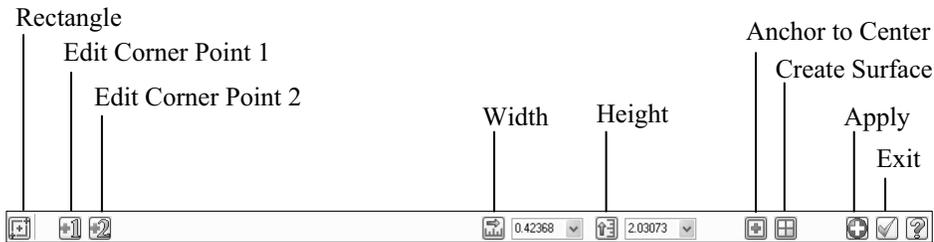


The rectangle ribbon bar can also be selected from the **Sketcher** toolbar.



The Rectangle option uses the ribbon bar shown below.

**Rectangle
Ribbon
Bar**



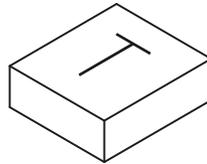
Item	Definition
Rectangle	Function Icon.
Edit Corner Point -1	Edit the first corner point.
Edit Corner Point -2	Edit the opposite corner from Point-1.
Width	Width (X) of rectangle.
Height	Height (Y) of rectangle.
Anchor to Center	Use center point of rectangle as anchor point. If this option is off, the cursor is set to select opposite corners of the box.
Create Surface	Create a trimmed surface bounded by the rectangle in addition to the wireframe geometry.
Apply and Stay	Create rectangle and stay in function.
Exit	Create rectangle and exit function.

Letters

There are two ways to create letters for machining in Mastercam.

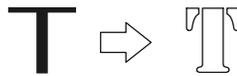
- Mastercam wireframe characters
- TrueType® Fonts

Wireframe characters, made up of lines, arcs and splines, are included with Mastercam. Of the wireframe characters, **MC9 (Box) Font** is widely considered the best for single-point machining part numbers and other simple text.



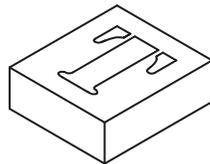
Single Point Box Font

TrueType® fonts are characters used in documents. Fonts use a kind of shorthand that saves computer processing time and memory. Mastercam automatically converts TrueType fonts to machinable curves.

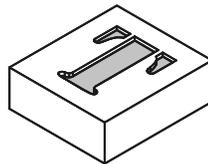


TrueType to Wireframe

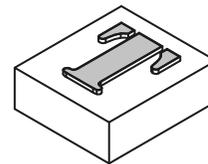
The outline can be single point machined but more often pocket toolpaths are used to create recessed or embossed letters. .



Single Point



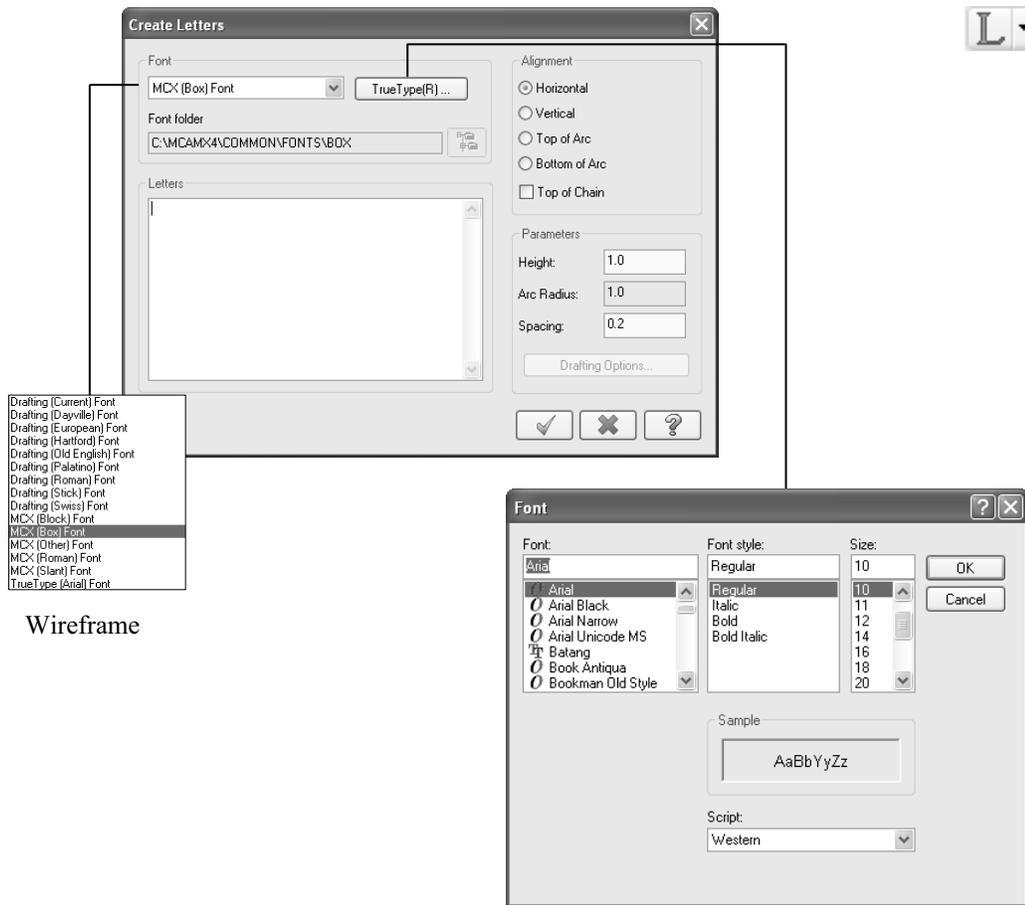
Recessed



Embossed

Create letters by selecting the **Create, Letters** option from the Menu.

Letter Options



Wireframe

Truetype Fonts

Item	Definition
Font	Select the font source and style.
Alignment	Choose direction and layout of text.
Parameters	Text size, spacing and radius (if wrapping along an arc).
Drafting Globals	Change active drafting text settings.

Bounding Box



Use **Create, Bounding Box** to create a cube or cylinder around selected entities. This function is especially useful for determining stock size.

Select Entities to Bound

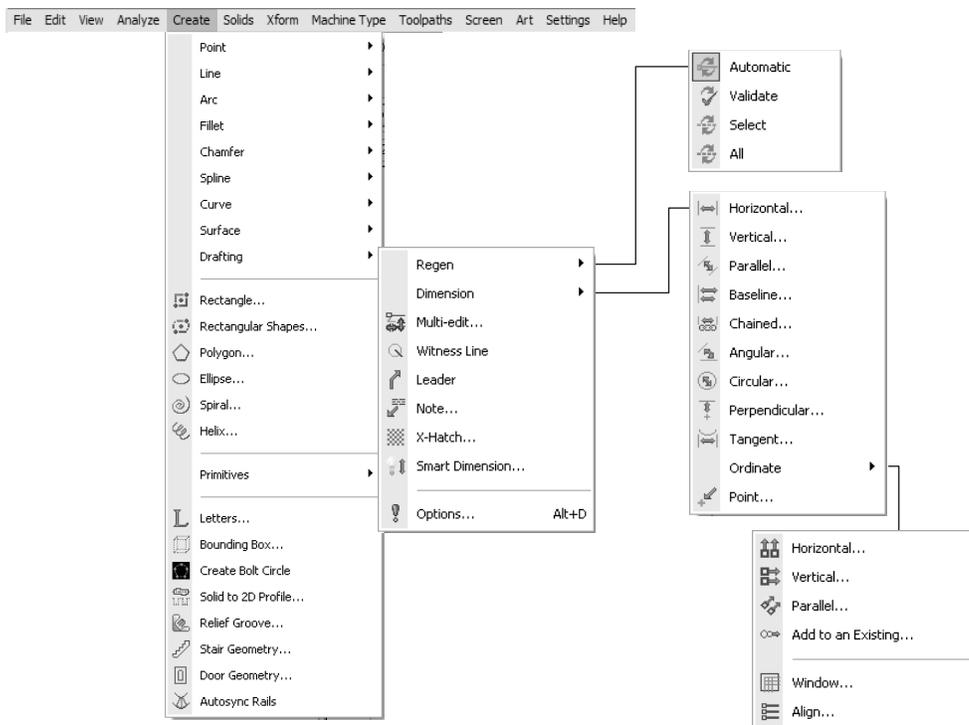


Item	Definition
All Entities	Box will be created around all visible entities. Use Select to choose the entities to form the box around.
Create	Determines what entities will be created. <ul style="list-style-type: none"> • Stock sets the stock boundary for toolpaths. • Lines Arcs creates a wireframe boundary. • Points creates a point at extends of boundary. • Center point creates a point at the center of the cube.
Expand	Amount to extend the box beyond the selected entities.
Shape	Select rectangular and cylindrical bounding.
Axis	Axis of revolution of a cylinder.

Mastercam has extensive drafting capabilities. Unless you are doing tool design, you will probably not create detailed drawings. However, some basic dimensioning skills are useful for creating setup sheets, documenting your work for future reference, or communicating your ideas to others.



Begin by choosing the **Create, Drafting** drop down menu or by choosing the drafting icon. To learn more about drafting functions, select **Alt-H** for Mastercam Help.

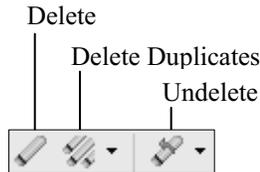


Item	Definition
Regen	Regenerate associative dimensions; dimensions that change as geometry changes.
Dimension	Select type of dimension to create.
Multi-edit	Change the drafting parameters of many dimensions at once.
Ordinate	Ordinate dimensions are in reference to a fixed datum point.
Options	Set default drafting parameters such as text height and styles.

Delete



Select **Edit, Delete** from the main menu, or select a Delete icon to access the delete functions.



Item	Definition
Delete	Delete selected entities.
Delete Duplicates	Duplicate entities are usually unnecessary and make chaining, or selecting connected entities, more difficult. This function tests all visible entities and deletes duplicates.
Undelete	Recovers deleted entities.

Here are some fast and easy ways to pick entities and delete them.

- **Pick Single Entities:** Click once with the left mouse button on the entity to select it. Hold down the Ctrl key to select multiple entities.
- **Window Entities:** Click and hold the left mouse button and drag the cursor to form a rectangle around them and then release the button.
- **Chain Select:** Hold down the Shift key and click on an entity to select

After selecting the entities, select the Delete key on the keyboard or the Delete icon to delete the entities.

Groups

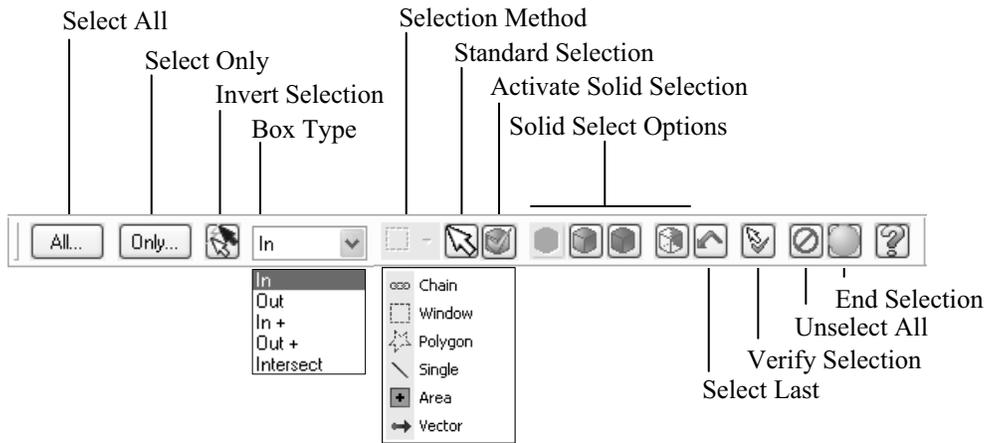
As you select entities, Mastercam temporarily adds the entities to a selection set, called a **group** and changes its color. Groups allow you to deal with entities together, rather than as individual pieces.

There are many ways to form a group other than those just listed and many reasons to create groups other than deleting them. For example, you may want to move, rotate, or change the color or level of many entities at once.

The previously described methods for forming a group works well if you need to pick just a few entities, if they form a simple chain, or if they can be selected using a window.

General Selection Tool Bar

Sometimes, none of these methods work well. In these cases, use the General **Selection Toolbar** to form and manage groups.



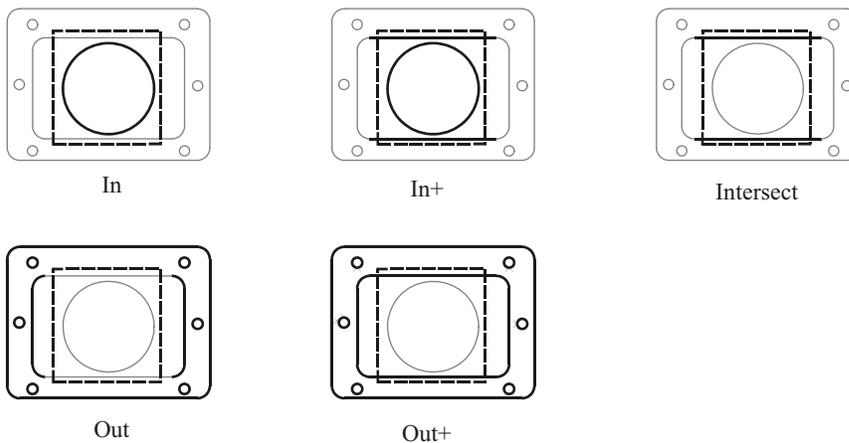
Box Type controls how a selection Window or Polygon behaves.

Item	Definition
Select All	Select all entities by attributes, type or other criteria.
Select Only	Set cursor to select only entities of a specified attribute, type, or other selection criteria.
Box Type	Sets how the selection window or polygon behaves.
Selection Mode Override	Override selection mode for a single pick.
Toggle Standard/Solids Mode	This option switches between standard and solid selection modes. Solid selection is covered in Chapter 5, Solid Modeling.
Verify Selection	Appears in graphic area to let user confirm each selection.
Unselect All	Empty the group selection.
End Selection	Finish group selection and proceed to next step.
Help	Help on this toolbar.

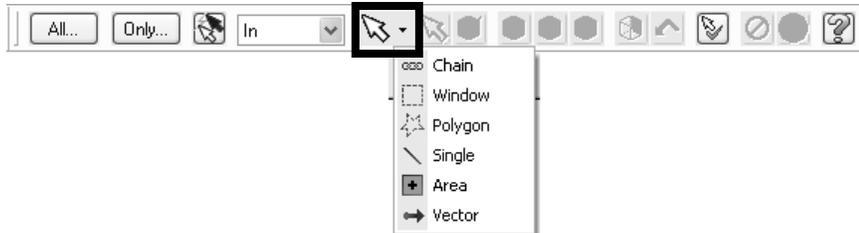


Item	Definition
In	Sets box so only entities completely within are selected.
Out	Sets box so only entities completely outside the box are selected.
In+	Sets box so only entities that are completely in or intersecting the box are selected.
Out+	Sets box so only entities completely outside or intersecting the box are selected.
Intersect	Sets box so only entities that intersect the box are selected.

The following diagram illustrates how each box type works. Selected entities are shown in black.



Selection Override options lets you use the cursor or combinations of the cursor and keystrokes to select entities.

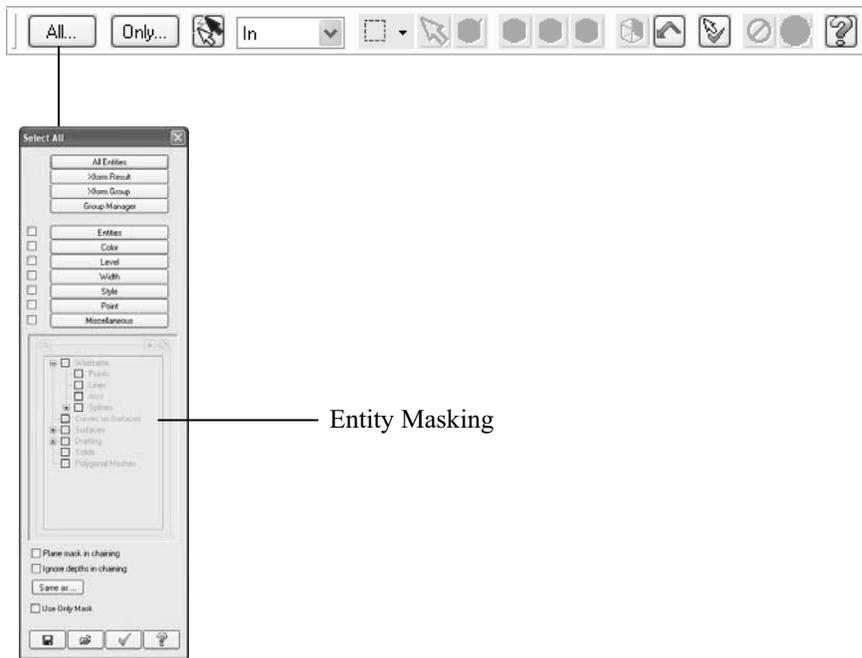


Item	Definition
Chain	<p>Select a connected profile of entities. To select a partial chain, pick the first and last entities in the profile that you want to select.</p> <p>To select multiple chains, select the first chain and then hold down the Ctrl and Shift keys for subsequent chains.</p> <p>Chaining is discussed in Chapter 7, Contour Toolpaths.</p>
Window	Allows selection using a rectangle. Click and hold the left mouse button, drag the mouse, and then release the button.
Polygon	Allows selection using a sketched profile. Press Enter or double-click the last point to end the polygon selection.
Single	Allows selecting a single entity. To remove entities from the group, click on them again.
Area	Allows selection of an area bounded by a chain. Click inside the outermost boundary, but outside the inside entities.
Vector	Picks any entity crossing a sketched line
Arrow	Revert to General Selection mode.

Select All

The **Select All** dialog box allows selecting of all entities of a specified type, attribute, diameter or length.

For example, to select all lines, click to place a checkmark by the **Entities** and **Lines** options and then click on the **Exit** option.

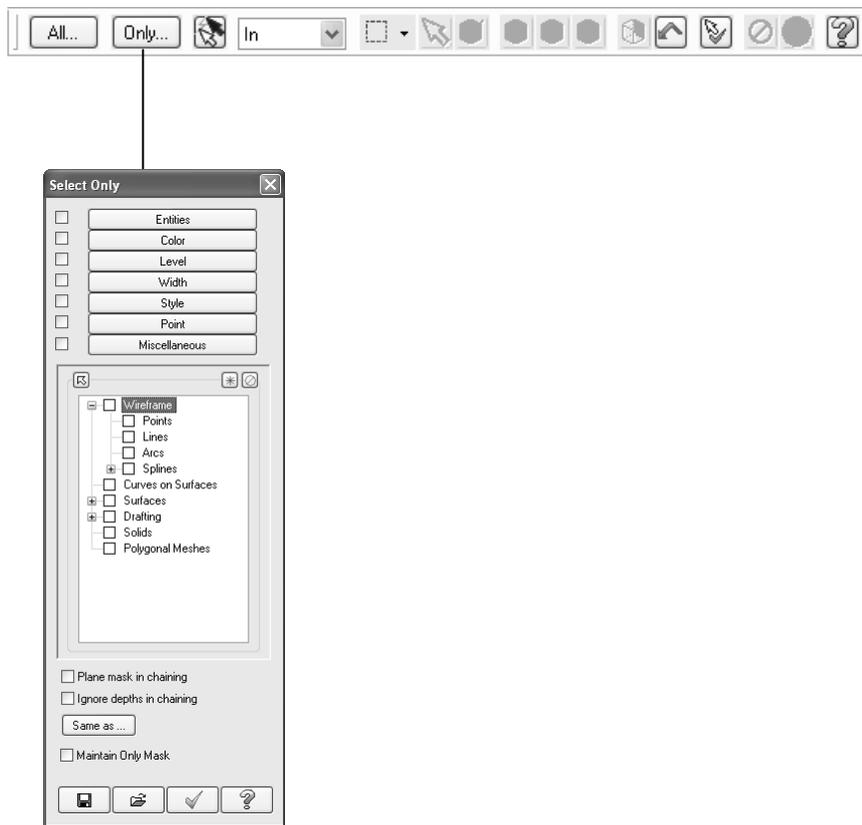


Item	Definition
Entity Masking	Refine Entities selection.
Plane mask in chaining	Select only entities on the same plane as the selected entity in a chain.
Ignore depths in chaining	Chain entities that are at different Z values but which share XY coordinate endpoints.
Same as...	Set the selection criteria the same as an existing entity.

The **Select Only** dialog box lets you configure the cursor to only select entities of a specified type, attribute diameter or length.

Select Only

For example, to select only arcs of a specified diameter, click to place a checkmark by the **Diameter/Length** option and the **Diameter** option. Then enter a value in the **Diameter** field. Exit the Select Only dialog box and pick entities from the graphic area.



Select
Only



Step by Step

The following shows how to select and delete arcs of a specified diameter.

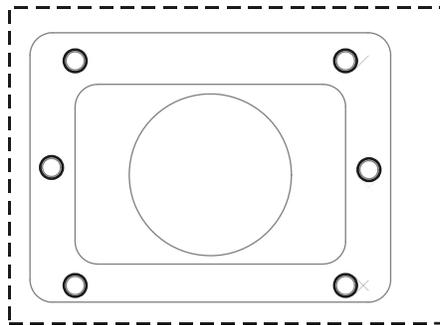
Step 1: Select **Delete** from the menu or Delete toolbar.

Step 2: Click the **Only** option on the general selection toolbar.

Step 3: Click on the **Diameter/Length** option.

Step 4: Click on the **Diameter** option and enter a value, and then Exit the dialog box.

Step 5: Use the cursor to sketch a window around the arcs to select.



Step 6: When done, click on the **End Selection** icon.



Power
User

Paste geometry from one Mastercam file to another using the Edit, Cut, Copy and Paste functions.

1. Open a second instance of Mastercam.
2. Select the geometry to be exported.
3. Select Edit, Copy (or Cut).
4. Move the mouse into the other Mastercam window and select Paste.
5. A Paste ribbon bar appears that allows changing the attributes of the geometry to the active attributes.
6. Select OK.

Mastercam Undo does not work for this function, so save your work before using it.

This chapter is intended to give you the basic computer knowledge and skills you need to begin using Mastercam.

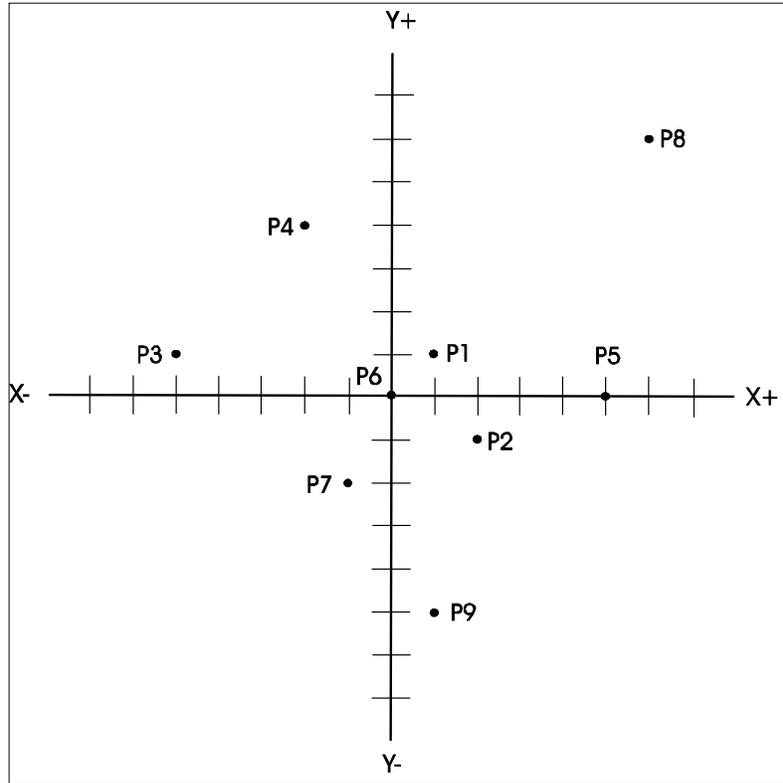
**What
You
Learned**

- Review of Cartesian coordinate system.
- Absolute vs. Incremental coordinates.
- Definition of Quadrant.
- Datum selection.
- Definitions of wireframe, surface and solid geometry.
- How to draw lines, arcs, points and rectangles.
- Using AutoCursor.
- Entering numeric data.
- Using the mouse and mouse buttons.
- Change viewpoints.
- Creating fillets and chamfers.
- Trimming, extending and breaking entities.
- Creating letters for machining.
- Dimensioning shop drawings.
- Using the select tool bar.
- Using the delete and undelete functions.
- Using the undo/redo functions.
- How to find and fix problems with chains.

Exercise 3-1: Cartesian Coordinate System

Write the **Absolute** Coordinates and Quadrant for Each Point

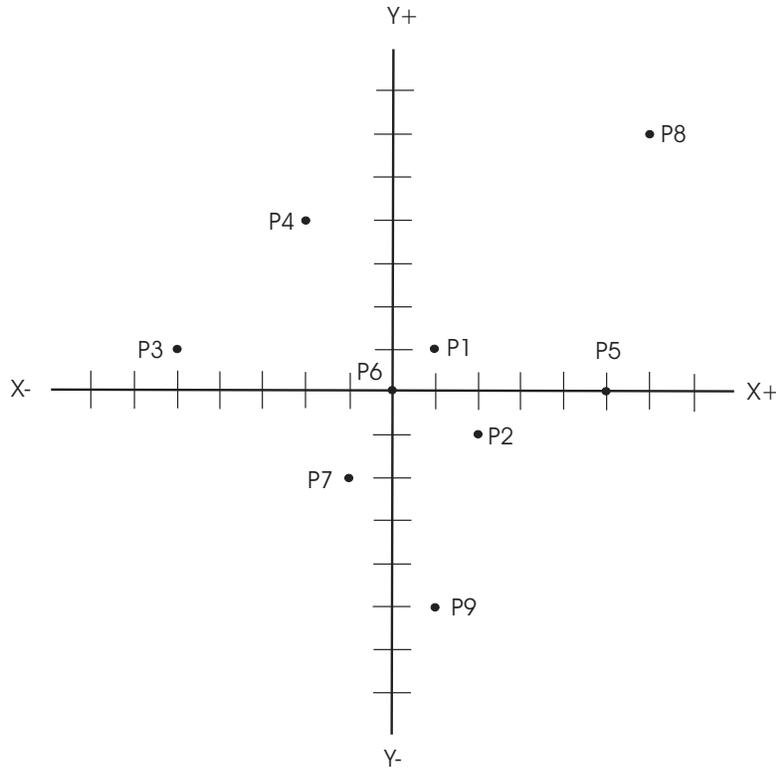
See solution on page 3-70.



Point	X-Value	Y-Value	Quadrant
P1	1	1	I
P2			
P3			
P4			
P5			
P6			
P7			
P8			
P9			

Exercise 3-2: Incremental Positioning

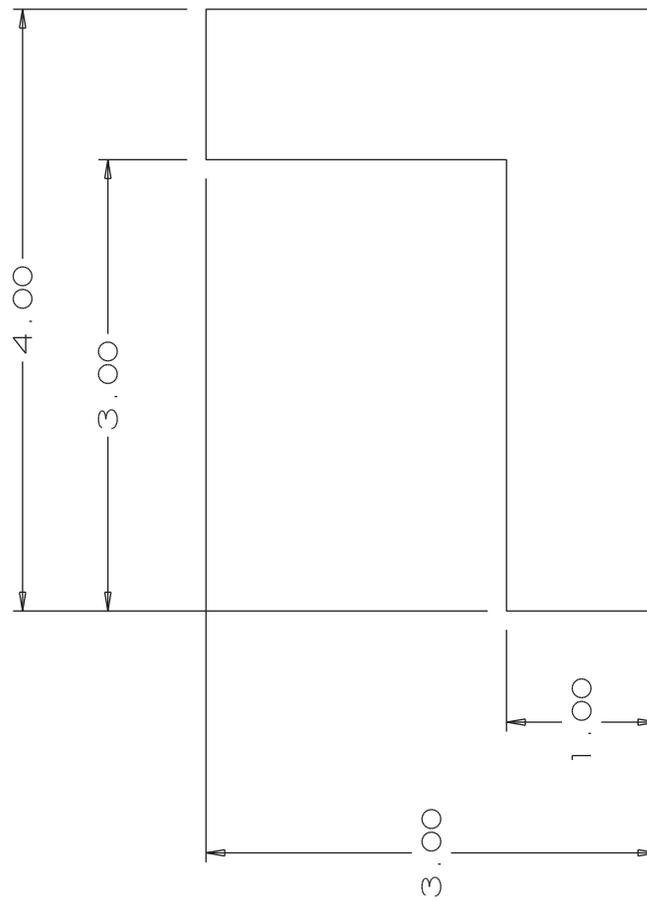
Write the **Incremental** value for moves between the points.
See solution on page 3-70.



Move	X-Value	Y-Value
Origin to P1	1	1
P1 to P2		
P2 to P3		
P3 to P4		
P4 to P5		
P5 to P6		
P6 to P7		
P7 to P8		
P8 to P9		

Exercise 3-3: L-Block

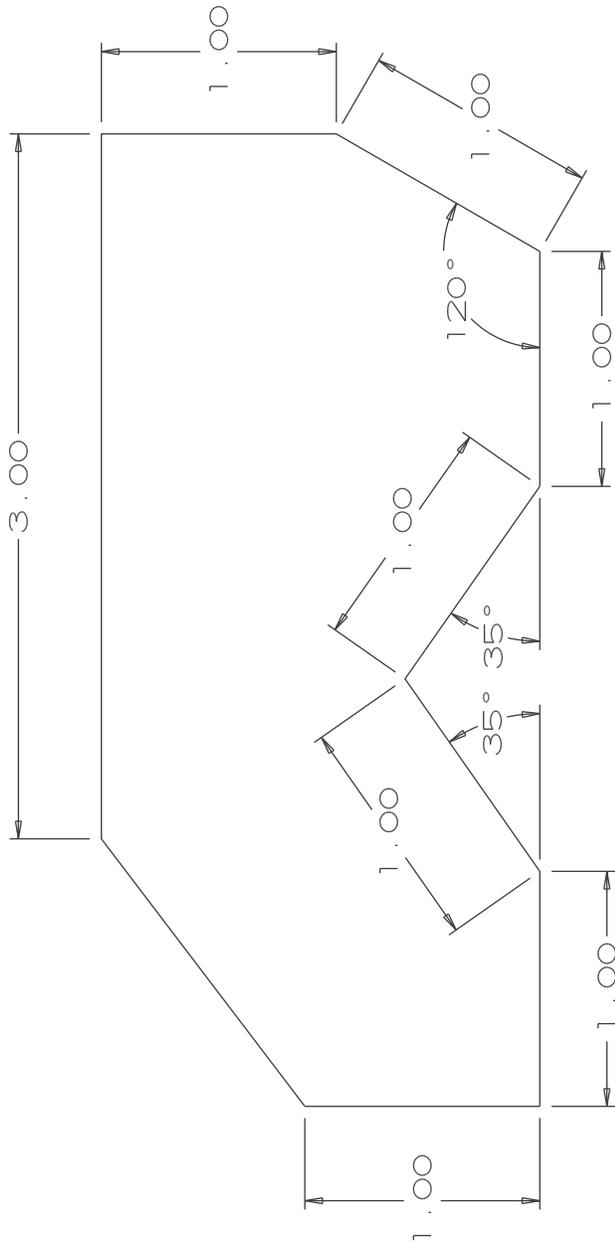
Draw the shop drawing below in the correct location and quadrant as shown. Do not dimension.



Shop Drawing
Not To Scale

Exercise 3-4: Polar Lines

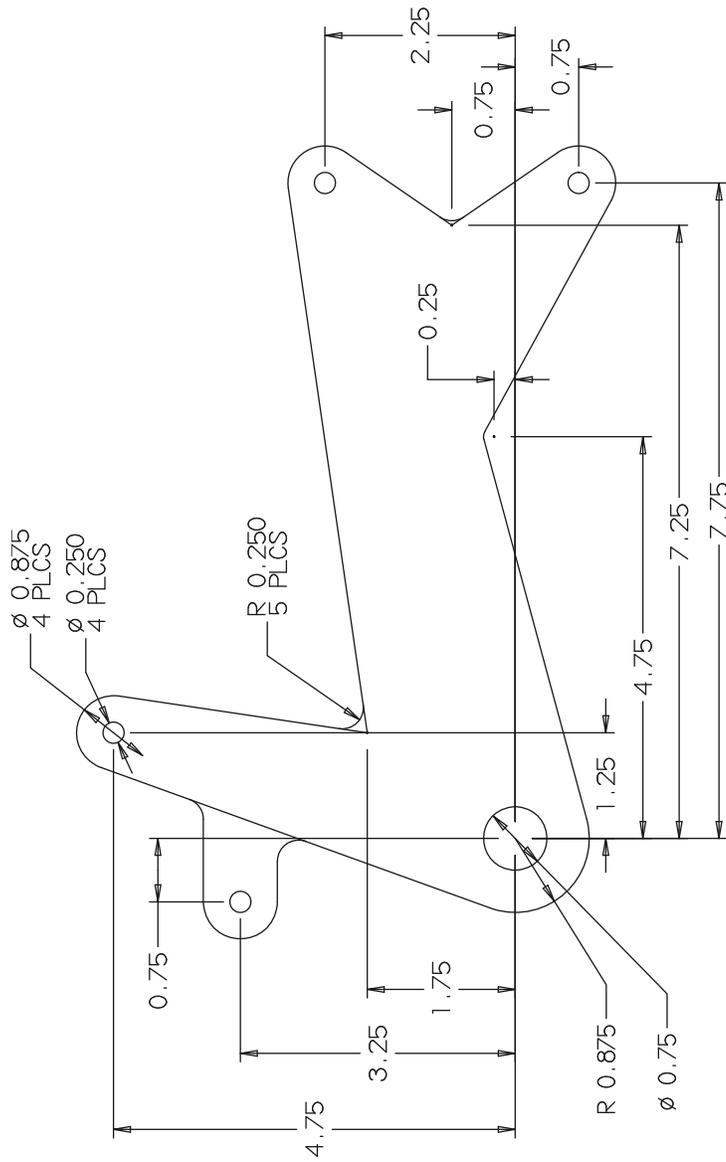
Draw the shop drawing below in the correct location and quadrant as shown. Do not dimension.



Shop Drawing
Not To Scale

Exercise 3-6: Control Horn

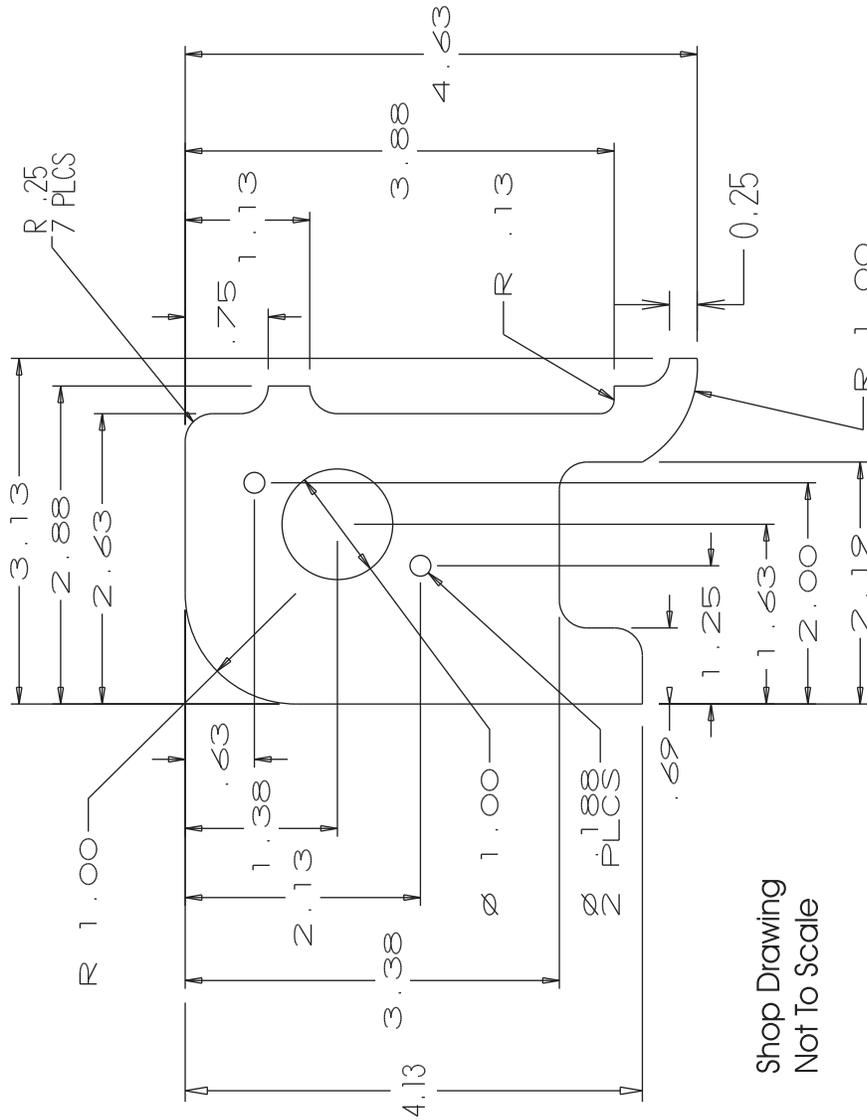
Draw the shop drawing below in the correct location and quadrant as shown. Do not dimension or draw phantom lines or points.



Shop Drawing
Not To Scale

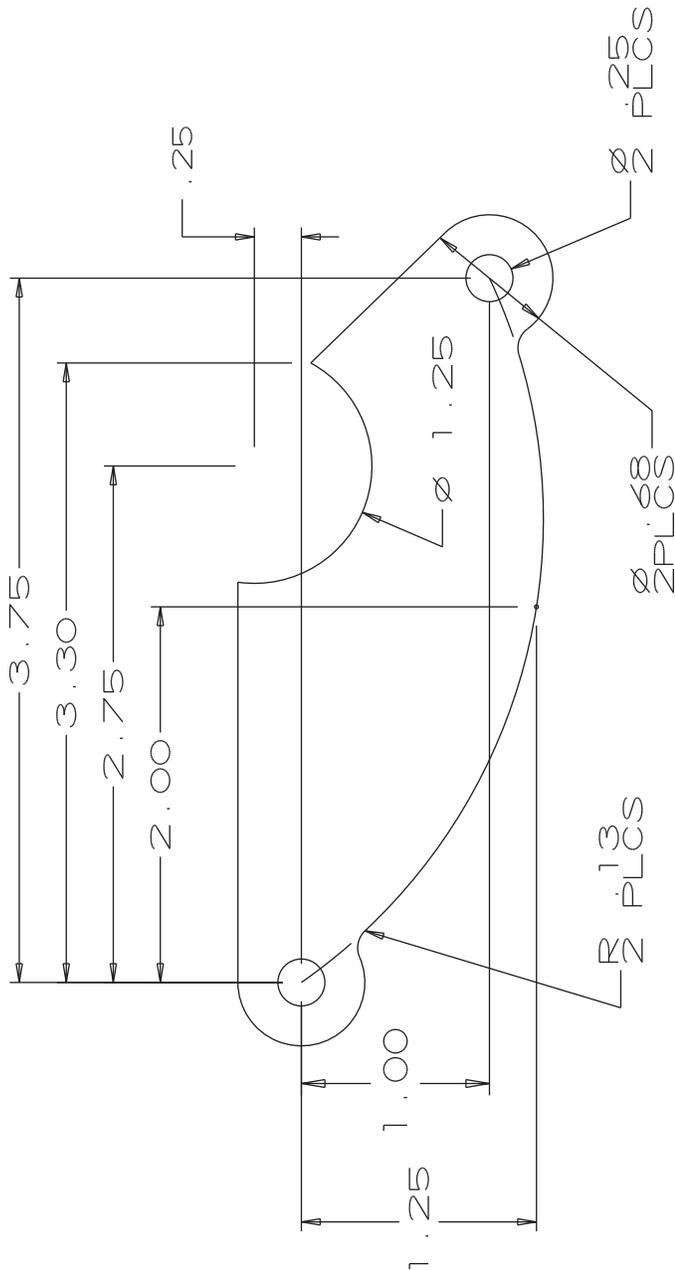
Exercise 3-7: Reinforcing Plate

Draw the shop drawing below in the correct location and quadrant as shown. Do not dimension.



Exercise 3-8: Bell Crank

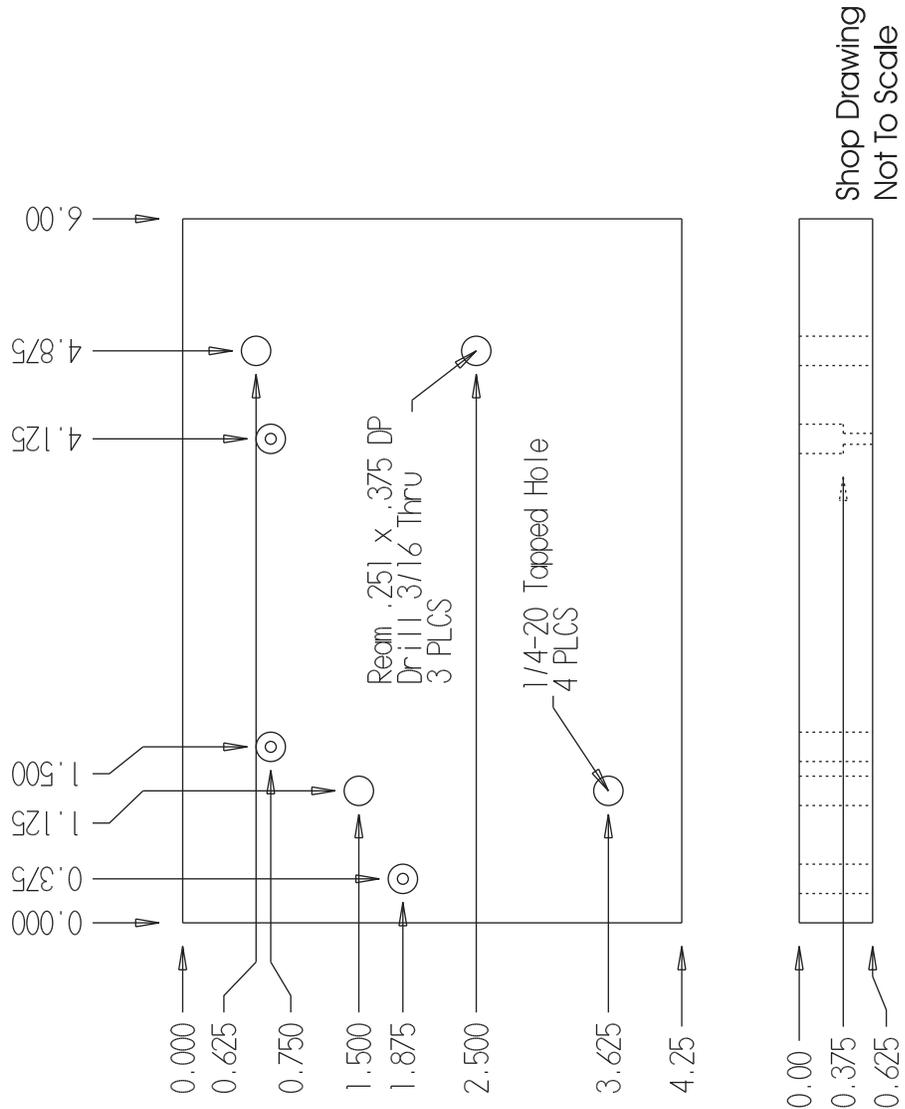
Draw the shop drawing below in the correct location and quadrant as shown. Do not dimension.



Shop Drawing
Not To Scale

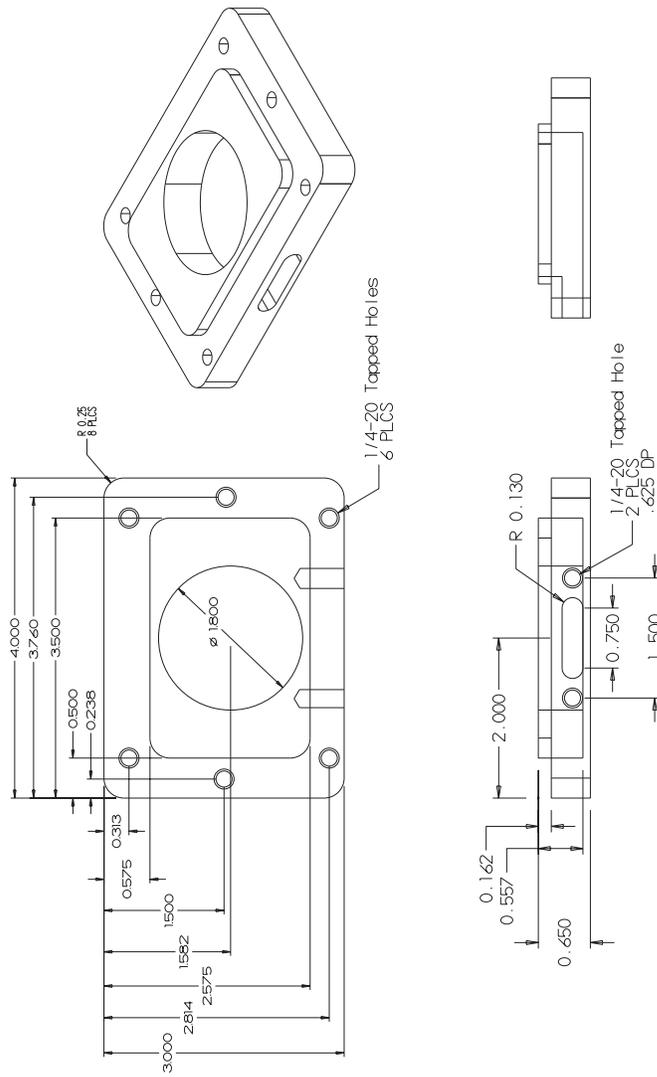
Exercise 3-9: Fixture Plate

Draw and dimension.



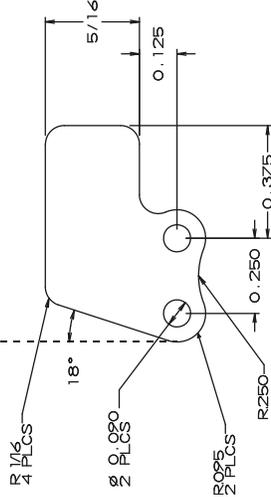
Exercise 3-10: Distributor Manifold

Draw and dimension. Do not draw ISO view.



Shop Drawing
Not To Scale

Exercise 3-11: Chain Breaker
Draw and dimension.



Shop Drawing
Not To Scale

Exercise 3-12: Wireframe Lettering

Create your name using the MC9 (Box) Font, 1" high

Exercise 3-13: TrueType Lettering

Create your name using the Times New Roman Font, 1" high

Exercise 3-14: Drafting Exercise #1

Dimension the drawing, L-Block

Exercise 3-15: Drafting Exercise #2

Dimension the drawing, Bell Crank

Solution to Exercise 3-1: Cartesian Coordinate System

Point	X-Value	Y-Value	Quadrant
P1	1.	1.	I
P2	2	-1	IV
P3	-5	1	II
P4	-2	4	II
P5	5	0	On +X Axis
P6	0	0	At Origin
P7	-1	-2	III
P8	6	6	I
P9	1	-5	IV

Solution to Exercise 3-2: Incremental Positioning

Move	X-Value	Y-Value
Origin to P1	1	1
P1 to P2	1	-2
P2 to P3	-7	2
P3 to P4	3	3
P4 to P5	7	-4
P5 to P6	-5	0
P6 to P7	-1	-2
P7 to P8	7	8
P8 to P9	-5	-11

Additional Exercise

This file is stored in PDF format on the student CD to be printed and used for additional drawing practice.

Name
Geneva Gear