Press brake controllers

Not all press brake controllers have the same functions; some are very basic and some very sophisticated. In order to learn about the press brake controller we need one to discuss. For this section I have chosen a 1990’s vintage Amada NC9 because the controller functions are all on one face.

Many modern controllers (circa 2013) find these functions buried across several screens and pages which is fine but would make the discussion hard to follow. It is not the intent to promote or distract from any particular brand of controller, rather, to explain those functions as simply as possible.

The functions that may require an input on this controller include: the Ram Depth, Backgauge Origin, Flange lengths, etc. Your controller functions and inputs will be described in some detail in the instruction booklet that comes with your press brake… review it!

There are differences between one machine and another. We will look at the controller functions only in general terms.

Remember: Do not give too much meaning to the way one manufacturer or another labels things; a “bend dimension” or “flange” are the same implied function.

Press Brake Axes

Figure 2 displays the various axes and their labels consistent with a modern press brake backgauge. Note, these axes can differ in name from those of the brake itself, e.g.: the “X” axis in figure 2 is the “Y axis of the backgauge shown in figure 1.

Read the operator’s manual before attempting to program or operate any press brake.

The Modes

There are three different modes in which you can operate most controllers: The Depth Mode, Angle and Graphic modes.

Skilled operators will argue back and forth as to which is the better way to go, but it comes down to which method any given operator is the most comfortable with.

Depth Mode runs from direct inputs; backgauge to X.XXX and depth of penetration X.XXX. Angle Mode can be characterized as the relinquishment of most control functions to the computer. Graphic mode is similar to angle mode except everything is displayed with pictures, including runtime graphics of the forming operations.

MDI or Depth Mode

Manual Data Input (MDI) is also referred to as “Depth Mode” and can best be described as the manual operation of the press brake with only a minimum of computer control. Only basic and direct program data is submitted to the controller. All that is necessary is the depth of penetration and backgauge location.

All other functions are called on an “as needed” basis. The program tells the controller to move the backgauge directly
to the stated location, and when the foot switch is depressed, the ram moves to the input location. No controller processing of the data is required.

**Angle and Graphics Modes**

The best way to describe “angle mode/Graphics Mode” is to say that you are going to allow the controller to assume complete control over the operation of the press brake, calculating from the bend data and the physical operation.

In Angle and Graphic modes large amounts of data are normally required for the program, more than just telling the controller to go to direct location as in Depth (MDI). Angle or Graphic modes require information to calculate with: desired bend angle, bend length, die opening, tensile strength of the material, etc.

Each time a particular station is called by the program, the controller “processes” the data and determines the appropriate course of action. **The controller does not save the data once processed**; it reprocesses the data every time it is called. This may, depending on the vintage of controller, limit some of the other controller functions.

For example the open-height will need to open further than necessary for depth mode just because of the processing time and the delay in signaling the drive mechanism to stop. This is a very small amount, but there is a difference.

The function labels that will be described here are going to vary from controller to controller. Most will have comparable features though they may be labeled differently. Again, you will need to consult the operator’s manual for the specific brand and model for the machine you are running.

**Referencing the Depth Origin**

The depth origin can be set one of three different ways. The first is the single axis press brake.

Single axis press brakes are those without Z axis control; these are set manually by adjusting the depth of penetration at the bottom of the stroke. This is the “depth origin” process for a mechanical press brake. Normally you would set the depth to an amount equal to or greater than the material thickness (Mt). This would ensure that you will not lock up the ram or break the tooling on your first bend. From that point the ram is adjusted up or down by hand to achieve the required angle.

The second way sets the depth for CNC machines and is a simple task. The tooling comes under increasing pressure from contact between the punch and die as it is brought up to a given pressure, this predetermined pressure is set by the manufacturer and is intended as a common zero point for the controller, figure 3.

It also allows every operator to find the same reference point, regardless of who wrote the program or the specific machine being used. Sometimes you need to use pieces of tooling too small to withstand the pressure of zeroing. When this problem arises, begin with enough tooling to achieve the required tonnage of zeroing; then remove the excess tooling. It is also common to find machines that zero on top of the material; at the pinch point and under a slight load, figure 4. Both will give you a consistency of processing that makes set-ups faster and easier.

Note: Should the tooling requirement have a maxim allowable tonnage of a half ton, but the press requires four tons to zero, extra tooling needs to be added to allow the extra tonnage required to origin the machine.
Referencing the backgauge

Regardless of which mode you prefer, MDI, Angle or Graphic, there are a few things they have in common. To start with, how is the press is zeroed and how is the backgauge origin is found.

To understand how the backgauge is calibrated you need to look at it the same way the computer sees things. In figure 5, left side, you see the orientation the way that we look at it, from a human point of view: with the backgauge sitting in the 4.000 inch position and with the zero point being located at the center of the die set, figure 5. The right side shows the computer’s point of view.

The computer sees the die set as 4.000 inches and the backgauge location as zero. Many modern press brakes use 4.000 inches as their zero point.

Note: 4.000 inch setting may vary from machine to machine.

When you push the backgauge origin button, the backgauge will find what it considers to be zero. At that point the controller knows the backgauge is zeroed, and the controller will be ready to accept the required backgauge dimension. When you input a dimension into the controller, it is automatically adjusted into the computer point of view.

Take, for example, figures 6 a & b, if the controller moves the backgauge to the 3.000 inch position, the backgauge will find what it sees as 1.000 inch.

There is a good reason to pay close attention to how the controller sees things. It may become necessary to adjust the backgauge position reference point to reflect a true and valid location. To make the backgauge origin move closer to the die set, you will need to input a positive number and move the backgauge away from 4.00-inches, and will require a negative input.

For example, you bent a part and it measures 4.010 inches instead of 4.000. All the other bends were also large by .010. To correct the problem using the backgauge origin you would need to input +.010 into the backgauge parameter.

Then when the “Origin” button is pressed and the backgauge re-zeroed, all of the steps in the bend will have been adjusted by .010. Please note that any change made to the reference point will remain in effect until the next origin set; regardless of the number of programs run in between.

For the older and less sophisticated variety of backgauges or for those folks that do not use precision ground tooling, there is a slightly different method by which the backgauge reference location is found. With standard American style planed tooling, each tooling change (piece or direction) will result in a change in center location. This shift of centers is covered in more detail in the press brake tooling chapters of this book.
Because the center of the tooling moves in relationship to backgauge, the backgauge reference point will need to move to match. This is accomplished through the use of a backgauge calibration gauge block.

Two gauge blocks are centered in the tooling under slight pressure, one at either end of the punch and die set. The backgauge beam is loosened from its clamps and manually slid against the gauge blocks. Extra care is needed to ensure backgauge “squareness” to the tool center, using only light and equal pressure. When the beam is re-clamped into position and the gauge blocks are removed, the backgauge is calibrated. Figures 7 and 8 show a typical gauge block and calibration process.

Moving the reference point

Regardless of the type or brand of controller there will be times when adjustments need to be made to the backgauge reference location. In order to make this type of adjustment most machines will require you to enter your data changes in the computer’s terms. For example, if you need the backgauge to move out .050-inches, you will tell the controller to move the backgauge .050-inches in a negative direction. Remember, the computer sees things opposite from you, figure 6.

The backgauge is moved in and out by “stepper motors” and can have either one or two motors. The arrow is pointing at the stepper motor in figure 9.

Stepper Motors

From time to time dual motor machines need to be brought into or taken out of square. The object is to obtain the same physical location on both right and left sides.

Start the press with the controller empty and begin by setting the parameters back to the factory specifications.

These are usually found written down inside the controller cabinet or the machine’s manual. Once the machine has been zeroed, install tooling on both sides of the press brake bed. Bend a test piece, one on each side of the press. Next adjust the parameters so the test pieces measure the same on both sides. Remember, the data may need to be changed into the computer’s terms.

(MDI) Depth Mode

Now, let’s discuss in detail the options specific to depth mode. Remember this is the mode where the controller requires the least amount of information to operate. Ram Depth function. You first need to input the depth of penetration. For the controller in our example this is the distance from zeroed location at the bottom of the die to the starting point; about 125% to 150% of the material thickness is the “best practice” for a 90° bend in a 18-inch piece of cold rolled steel.

For example, take a six inch piece of material with a thickness of .060-inches, a good place to start would be a depth of .075 to .085-inches above the point where the die faces touch without the material present.

This is assuming you are “air forming”. You must remember that this number will go up or down in direct relationship to the bend length.

Note: *The dimensions listed for depth are from a zero point at the bottom of the V-die up!*
The Flange Function

Next is the flange length: how far back the backgauge located form the center of the tool. When running in depth mode there are two different ways to input the data. The first is to input the outside dimension of the bend and then use the bend allowance function to compensate for the elongation; or, you could calculate the location of the bend line and input that as a direct backgauge dimension.

The Bend Allowance Function

When adjusting for changes in dimension that occur during the bending process, the controller’s “Bend Allowance” function is used. It is used to compensate for the changes in length from bending; the bend deduction or to adjust a dimension.

A “bend allowance (BA)” used in calculating bend deductions (BD) is not the same as the “bend allowance” used on many controllers; this function may also be labeled: setback, bend deduction, etc. Note, those labels will change from brand to brand. Caution is necessary as these labels on the controller are not necessarily the same as those functions described in the Dissecting Bend Deductions chapter. In reality, a controller’s bend allowance function is equal to one half of a bend deduction (BD) as described previously.

Ram Depth

Flange Location Function and Bend Allowance (BA) are the three basic pieces of information required to operate the Press Brake in MDI mode. Even though you’re running a CNC controller, “depth mode” is similar in many ways to running an older style, non-CNC Press Brake. Only now you have the advantage of multiple steps, extreme accuracy and the repeatability of a CNC control.

The next levels of controller function are cross-over functions. By that I mean they are intended to work in either “depth”, “angle” or “Graphic” modes. On our NC9 controller we’re using for the descriptions in this chapter, the first of these is the gripper height. This is the location where the actual back stop sits. Some modern controllers offer from three to nine positions: three standard positions and up to six variations of “flip up, move to position, gauge drop into place”. These are used to maintain a stop position in relationship to the tooling. This location setting might also be a direct input dimension; check your manual.

Some of the older or less sophisticated controllers will not have this function. The machine stops are raised or lowered by means of a hand screw.

Retract Depth Function

Retract depth or pinch point are the samething, where the punch grabs the material just hard enough to hold it while the backgauge retracts.

Retract length

[Retract length] is the distance that the backgauge will be moved out of the way. These functions are most often used to allow a reverse flange bend to pass. A reverse flange bend is one where the first bend is bent in the opposite direction in relationship to the second bend, down in this case. As this second bend is started, the first bend binds up on the backgauge. To avoid this binding the retract and retract depth are invoked.

The Lower Limit Function

The lower limit is used to control how far open the press will come from the zero depth. It allows different sized flanges to “pass by” the tooling when being removed from the brake. This is the “open height” and being able to adjust it step by step makes the press more efficient and a safer machine to operate. Because the machine needs to be opened only far enough for the part to be removed, the chances of a part of your body being included in the next bend...
are greatly reduced.

*Slow Bend Speed Function*

This is the rate that the ram actually pushes the material into the V-die space. This is not a required function but it does have many uses. It is used where back bending is an issue or when the operator needs delicate control over the operation.

*Slow Bend Position*

Just as you might have assumed, this is the point in the ram's approach that the “slow bend speed” function is activated.

*Angle / Graphic Modes*

The best way to describe “Angle or Graphic mode” is to say that you are going to allow the controller to assume complete control over the operation of the press brake, the calculating of all the bend data, and the physical operations. Graphic Mode just adds pictures to an Angle Mode program.

*The Angle Function*

The “angle” data input is equal to the final bend angle you wish to achieve after forming. The tricky part is that the angle input in some controllers is input as an included angle. This is the angle as measured on the inside. For example, a 45° bend would be input as 135°; a 92° bend would be 88°. And some are complementary angle inputs. The controller will compute the depth of penetration for the ram to achieve the desired angle.

Most controllers will have an angle adjust step. In angle mode the adjustment input needs its value in degrees. For this function the input will be a direct + or – decimal degree, changing the ram location up or down as necessary.

*The Bend Length Function*

In order for the controller to calculate how much pressure/depth is required, several pieces of information are necessary. One of those is the length of the bend to be formed. This is the total length of tooling that comes in contact with the workpiece.

*The Die Width Function*

The die width is one more of those required pieces of information used in the depth calculations. This is the measured distance across the die opening. It is used in the computing of tonnage.

*The Thickness Function*

This is pretty self explanatory; it the physical measured thickness of the material to be formed.

*The Tensile Strength Function*

Tensile strength can be defined as the material’s ability to bear weight without breaking or being pulled apart under a smooth load. The controller uses this data in the computing of the depth for punch penetration.

*The Punch Radius Function*

On the tip of the punch is a radius. This radius is what the controller needs to know for its bending computations. There is a unique rule that applies to this controller function. The input will not only depend on the punch tip radius, but also on whether or not it is a sharp bend. If the punch radius is sharp in relationship to the material thickness (Mt), the sharp bend value is what is need if the controller is to be expected to perform at its best.
The Die Radius Function

On the top two edges of the opening is a radius. This radius will vary from .008 to .125 or more. In order for the controller to compute the necessary accurately this radius must be incorporated. The sharper this radius is, the greater the amount of drag on the material as it is drawn into the die space. Each radius has a place and a purpose in the forming process; the controller just needs to know what you have selected to use.

The Die Angle Function

The included die angle is selected on the basis of the springback in the material or the radius/material-thickness relationship. Springback varies from one material to the next, the greater the springback is, the smaller the included die angle should be. The controller needs to know this angle to complete the mathematics necessary to run a part. Many controllers will have fewer functions and some will have more. This will depend on the sophistication of the controller and the number of axes being manipulated. An axis is a controlled action, i.e.: ram depth is one, backgauge two, stop height could be three.

Again, the functions we have just covered are based on the AMADA NC9, although it could have been any brand this particular model had all of the functions on the front interface rather than buried in menus.

Press Brakes and Methods courtesy of Asma LLC looks at the machine, methods and controls.

2 Press Brakes and methods

Press Brakes and Methods

Courtesy of Asma LLC