

Siemens Step 7

S7-1500

Programming

Table of Contents

	Page
A. Starting a Project With the S7-1500	Step7_1500-3
1. Rack Configuration	Step7_1500-4
2. Tag Table	Step7_1500-7
3. Configuring Communications	Step7_1500-7
4. Specifying Other Tags	Step7_1500-7
5. Processor Download	Step7_1500-8
B. Ladder Logic Programming	Step7_1500-9
1. Two Contacts in Series (Logical AND)	Step7_1500-9
2. Entering Branches (Logical OR)	Step7_1500-10
3. Timers	Step7_1500-10
4. Counters	Step7_1500-11
C. Adding Comments to Ladder Logic	Step7_1500-12
D. Assigning Symbols to Addresses	Step7_1500-12
E. Displaying Memory Locations	Step7_1500-13
F. Input/Output Forcing	Step7_1500-13
G. Importing/Exporting Symbols	Step7_1500-14
H. Saving Ladder Logic Program	Step7_1500-15
I. Restoring Ladder Logic Program	Step7_1500-15
J. Printing Ladder Logic Program	Step7_1500-16

This document is a supplement to the “Lab 1.doc” lab exercise that introduces one to the basics of ladder logic programming. The following equipment is assumed:

- Siemens 6EP 1333-4BA00 PM190 5A power supply
- Siemens 6ES7 516-3AN00-0AB0 CPU 1516-3 PN/DP processor
- Siemens 6ES7 521-1BL00-0AB0 32 chan. 24v disc. input
- Siemens 6ES7 522-1BL00-0AB0 16 chan. 24v disc. out/0.5A

This document is primarily for version 17 of the Step7 TIA Portal software.

In order to become familiar with basic ladder logic programming and addressing techniques on the Siemens equipment, the steps to accomplish the first part of the first lab exercise are outlined in sections A and B. The remainder of the sections outline the steps to document your ladder, to print the ladder logic program, and to save your ladder logic diagram.

In an industrial setting, the initial PLC programming is done offline and then later downloaded to the PLC when ready for the final test and debug. In addition, there is usually only one project (program) for each PLC on the network. For these labs, the ladder programming will be done online so that changes can be immediately tested.

Siemens Step7 TIA Portal version 17 is used to program the S7-300/1500 PLCs. An icon for these programs should already be on the desktop.

The PLC and the PC used to program it are connected through Ethernet.

A. STARTING A PROJECT WITH THE S7-1500

Wire the switch/light board before proceeding. Power up the computer, the PLC rack, and external power supply, as needed. The switch on the front panel of the PLC processor should be in the RUN position.

Make sure the computer, PLC, and external power supply are all powered.

Start the Step 7 software by clicking on the TIA Portal icon on the desktop, or by selecting

Start Icon | Siemens Automation | TIA Portal V17

It will take a few seconds to start.

You will probably find it easier to navigate the project with the Step7 software in the "project" view. Even if one starts in the "portal" view, the software switches to the "project" view anyway. If the "portal" view appears with major tasks on the two side bars, click on the **Project View** link on the bottom left of the screen.

Note: to make sure your project always starts in the "project" view, on the top menu bar, click on **Options** and then **Settings**. A settings window appears in the center of the screen. Scroll down to the "Start view" section. Select the **Project view** radio button. Close the settings window by clicking on the "X" in the upper right corner of the settings window.

If your project is already in the PLC, click the "open file" icon (manila folder with green arrow) and select your project from the list of projects (or browse if it is not listed), and then click on **OK**.

To start a new project, click on the "New project" icon (white paper sheet with yellow star, beneath "Project" menu item, or select **Project** and then **New**.

Type a name for your project and change the Path as needed. The

C:\Projects\Siemens folder is open to all users, but you will probably want to save it to your S: drive

Click on **Create**

In the "Project Tree" window on the left, a line with the name of your project should appear and the name of the project is shown in the title bar of the project window.

In the project tree, Double-click on **Add new device**

If the tree in the middle pane does not start with "Controllers" on the top line, click on the **Controllers** button

Expand the tree to get to the appropriate processor (make sure you get the correct one):

For both Rolla and Springfield, the processor is a CPU 1516-3 PN/DP:

SIMATIC S7-1500

CPU

CPU 1516-3 PN/DP

6ES7 516-3AN00-0AB0

Check that the version is 1.8. If not correct, select the correct version.

Give it a name (in box below "Device name:")

Click on the **OK** button.

The hardware is displayed in the center pane.
Click ***Save Project***.

When starting a project for the first time, it must initially be set up offline, and then go online. The initial setup consists of specifying the processor and configuring the discrete I/O modules.

You are now ready to configure the processor and I/O modules.

1. Rack Configuration

Using the Step 7 software, configuration of the inputs and outputs involves two steps:

1. specifying modules in the rack
2. assigning tags to the I/O channels

The second step is not absolutely necessary, but it is easier to specify tags now than when programming the ladder logic. The Step 7 programming software aids the programming of a PLC because one does not need to memorize the I/O addresses and file addresses, but can refer to them symbolically, as a tag.

The PLC will be set up so that the module configuration is as follows:

Slot	Module	Order Number	Firmware	Start Addr
0	PM190 5A	6EP1333-4BA00		
1	CPU 1516-3 PN/DP	6ES7 516-3AN00-0AB0	V1.8	
2	32 chan. 24v disc. in	6ES7 521-1BL00-0AB0	V2.0	%I0
3	32 chan. 24v disc. out	6ES7 522-1BL00-0AB0 Or 6ES7 522-1BL01-0AB0	V1.0 V1.0	%Q4

The hardware should appear in the center pane.

If the hardware does not appear in the center pane, in the project tree pane (left side), expand the line with the name of your PLC (followed by “[CPU xxx]”). Double-click on **Device Configuration**.

To configure the rack containing the processor, do the following:

The window displays a rack with the CPU already in slot 1.

If a right pane, displaying a catalog of available items, is not available, click on the right side-tab **Hardware Catalog**.

To insert a module into the configuration, the module is selected from the catalog in the right pane.

First insert the power supply. Expand the **PM** folder, and the **PM 190W 120/230VAC** folder. Select the appropriate module (6EP1333-4BA00).

Double-click on the entry in the hardware catalog. It should be placed in slot 0.

Alternatively, one may drag-and-drop the module to slot 0 of the rack displayed on the left.

To add the two I/O modules, do the following:

In the catalog, Expand the **DI** folder and then the **DI32x24VDC HF** folder. Select the appropriate module (6ES7 521-1BL00-0AB0).

Double-click on the entry in the hardware catalog. It should be placed in slot 2.

Alternatively, drag-and-drop the module to slot 2 of the rack displayed on the lower left pane. It should appear in slot 2 and its “I” address should appear in its row in the Device Overview pane (to the right of the rack graphic. May need to scroll horizontal) and be automatically assigned to “0..3”.

The version needs to be changed to 2.0 (the default is version 2.2). Right-click on the module in slot 2 and select **Change device**. On the right side of the pop-up window, expand the **DI32x24VDC HF** folder. Select the appropriate module (6ES7 521-1BL00-0AB0). Make sure the Version on the "New device" side of the window is "V2.0". If not, click on the down arrow and select "V2.0". Click on **OK**.

If the "I" address is not "0..3", change the start address of the discrete output module by double-clicking on the row in the Device Overview pane and change the entry in the I address column to “0..3” by changing the first digit.

Expand the **DQ** folder. Check the catalog number on the discrete output module (front – top). If a 6ES7 522-1BL01-0AB0, expand the **DQ32x24VDC/0.5A HF** folder and select the appropriate module (6ES7 522-1BL01-0AB0). If a 6ES7 522-1BL00-0AB0, expand the **DQ32x24VDC/0.5A ST** folder and select the appropriate module (6ES7 522-1BL00-0AB0).

Double-click on the entry in the hardware catalog. It should be placed in slot 3.

Alternatively, drag-and-drop the module to slot 3 of the rack displayed on the left.

It should appear in slot 3 and the Q addresses should be automatically assigned.

If using a 6ES7 522-1BL01-0AB0, the version needs to be changed to 1.0 (the default is version 1.1). Right-click on the module in slot 3 and select **Change device**. On the right side of the pop-up window, expand the **DQ32x24VDC/0.5A HF** folder. Select the appropriate module (6ES7 522-1BL01-0AB0). Make sure the Version on the "New device" side of the window is "V1.0". If not, click on the down arrow and select "V1.0". Click on **OK**.

If the "Q" address is not "4..7", change the start address of the discrete output module by double-clicking on the row in the Device Overview pane and change the entry in the Q address column to "4..7" (only change the first number).

To configure the Ethernet interfaces on the processor, do the following:

Double-click on the darker grey rectangle in middle of the processor module and then click on the **Properties** tab in the lower part of the center pane.

In the lower center pane, click on the **PROFINET interface [X1]** selection in the tree on the left.

Scroll down to the **IP protocol** part of the properties

Set the properties as follows:

IP Address: 192.168.0.1

Subnet mask: 255.255.255.0

In the lower center pane, click on the **PROFINET interface [X2]** selection in the tree on the left. Scroll down to the **Interface networked with** part of the properties.

Click on **Add new subnet**. It should add a subnet of "PN/IE_1"

Scroll down to the **Internet protocol version 4 (IPv4)** part of the properties and set the properties as follows:

IP Address: whatever IT assigns

Subnet mask: whatever IT assigns

Use IP Router (dot in button)

Router address: whatever IT assigns

Go back to the PROFINET interface [X1] selection in the tree on the left. Scroll down to the **Interface networked with** part of the properties. Click on **Add new subnet**.

It should add a subnet of "PN/IE_2"

Set time zone. Click on the Time of day section in the tree on the left.

Time zone should be "UTC-06:00 Central Standard Time"

Daylight savings time changeover should be selected

Start of daylight saving time should be the Second Sunday of March 2:00 am

Start of standard time should be the First Sunday of November 2:00 am

Click on the Save Project icon (floppy disk).

2. Tag Table

To assign tags (names) to the input/output channels, select the “PLC tags” part of the project. In the project tree pane (left side), expand the line with the name of your PLC (followed by “[CPU 1516-3 PN/DP]”). Expand **PLC tags** and double-click on **Default tag table [0]**. A tag table with no tags will appear. To assign tags to the discrete input channels,

Click on the **<Add new>** in the “Name” column of the first row.

Type "Start_PB" and press <Tab>.

The data type will be automatically assigned as “Bool” and the address column will now have "%I0.0"

Press <Tab> 5 times to place the cursor in the Comment column.

Type in a description, for example, "Start push button switch" and then press <Tab> <Tab>.

Enter the other discrete input tags. Note that the address automatically increments to the next one.

Repeat the above 5 steps to specify the tags and descriptions for the other input and output channels. Note that when entering the first discrete output tag, the address will need to be changed to "%Q4.0."

To close the tag editor, click on the "X" in the upper right corner of the center pane.

3. Configuring Communications

Set up project communications. In the project tree pane (left side), make sure line with the name of your PLC (followed by “[CPU 1516-3 PN/DP]” is highlighted. If it is not highlighted, click on it. On the top menu bar, select **Online | Go Online**. Be patient, it will take a few seconds.

Set up Ethernet communications:

Type of the PG/PC interface: **PN/IE**

PG/PC interface: Some kind of Ethernet adapter (**do not select** PLCSIM)

Connection to subnet: PN/IE_1 (**do not select** local)

Make sure **Show devices with the same address** appears in a box above the table. If not, click in the arrow on the right side and make the proper selection.

Click on the **Start Search** or **Refresh** button.

After a few seconds, it should find the device and its IP address should appear in the “Address” column.

Click on **Go online**.

A popup may appear, asking, "Do you want to save these settings as default for the PG/PC interface?" Answer "No".

The top bar of whichever window pane is active should change to orange, indicating that you are online with the PLC

4. Specifying Other Tags

It is good practice to enter all of the tags before you start programming. If you enter a rung with undefined tags, errors will be reported. At that point, you can enter new tags into the program and clear the verify errors.

However, since timers and counters are handled quite differently than other PLCs, they will be defined as they are used.

To check that the processor is properly configured, the blank program should be downloaded to the processor.

5. Processor Download

If you followed the step above, you should be online (indicated by the orange top bar of the window in the center of the screen. If not already online, on the top menu bar, select **Online** | **Go Online**. The top bar of the window in the center of the screen changes to orange. If not, repeat the steps in part 3 above.

To download to the processor,

Right-click on the name of your PLC (followed by “[CPU 1516-3 PN/DPI]”).

Select **Download to device** and then **Software (only changes)**

If a “synchronization” window appears with blue and orange circles, click on the “Continue without synchronization” button.

A load preview window will appear. If there is a message indicating that the modules are stopped for downloading, then click on the arrow next to the pink box, and select “Stop All”. That must be selected to load the program.

Check for any other pink action boxes.

If the message indicates that there are differences between the hardware modules in the project and the modules in the target (actual PLC), you have incorrectly entered the hardware configuration. Click on the box on the lower bar with your processor name and verify the correct catalog numbers for all modules and verify the Ethernet configuration.

If the message is "Different project version in the offline configured device and target," it means the processor is at an earlier Portal version. Check the box to upgrade the target device.

Ask for help as needed.

Click on **Load**.

If a screen pops up with a message with a “Start module” action, click on **Finish**.

The next section, section B starts programming.

B. LADDER LOGIC PROGRAMMING

The steps necessary to program the first part of the experiment, the series rung, will be explained in detail. For the other parts of the experiment, only the different steps required to do these parts are explained.

1. Two Contacts in Series (Logical AND)

The first ladder program to be implemented will be a simple series control rung. Two switches, SW3 and SW4, will be placed in series and will be used to control a single light, LA1.

To show the ladder program in the Editor window, double-click on "OB1" in the left pane of the project window. Each ladder rung is called a network.

To put in the contacts and coil on the first network, do the following:

Click on the horizontal bar that is the blank network 1.

A normally open (NO) relay contact is needed at the input, so click on the appropriate contact symbol (-| |-) above the ladder window. A contact will appear on the left side of the rung with a red "???" above it..

Click on the appropriate contact symbol (-| |-) above the ladder window to add the second contact in series.

Click on the coil symbol (-()-) to add the coil to the rung.

Click on the "???" above the first contact and an edit line will popup.

Switch SW3 will be used first, so type the variable or address for that switch ("SW3" or "I0.2") and press <Enter>. The variable that controls the contact will appear above the contact symbol on the ladder diagram. Alternatively, if you have forgotten the variable name, after typing the first character, you may select the variable from the displayed list.

Note: the ladder editor can switch between displaying the symbol or the address by clicking on the "tag" icon (just to the right of the Download icon).

Click on the "???" over the other NO contact. Type the symbol or address for that switch ("SW4" or "I0.3") and press <Enter>.

Click on the "???" over the output coil and type the symbol or address for light LA1 ("LA1" or "Q4.0").

The online/offline box at the bottom of the window should display "offline." Download the changed program by clicking on the Download icon. If a red "STOP" is displayed at the bottom of the screen, start the processor with the following steps:

From the main menu, select PLC | Operating Mode

Click on the "Warm Restart" button.

Answer "Yes" to the next prompt.

Click on "Close".

On the bottom of the window, there should be a green bar "moving" right.

Click on the "eyeglasses" icon to turn on the program monitor.

Try the input switches in various combinations and observe the output. Also observe the changing status of ladder components on the screen. Energized or true conditions cause the elements to be highlighted (green) on the ladder diagram.

2. Entering Branches (Logical OR)

Click on the "eyeglasses" icon to turn off the program monitor. It is not necessary to stop the controller. All edits are made off-line and downloaded to the controller.

The SW4 contact is deleted and then added to the row below the contact for SW3. To place the SW4 contact in parallel with the SW3 contact,

Delete the SW4 contact by selecting it (it turns dark green) and pressing the key.

Now, click on the horizontal line to the left of the SW3 contact to highlight it. Open the branch by clicking on the icon that appears as an "L" with an arrow pointing right.

Now click on the NO contact icon to add the parallel contact.

Close the branch by clicking on the icon that appears as an arrow pointing right and up (next to the Open Branch icon).

Click the "???" above the new contact and type "SW4" to add the variable to it.

Download the program and start the processor. The controller is now set to respond to input conditions and perform appropriate output commands.

Note: to make on-line changes and download the processor, it is not necessary to stop the processor. If you download the processor while it is running, you will be asked if you really want to do the download, but you will not be forced to stop the processor.

3. Timers

A new blank rung is added to the program by clicking on the New Rung icon, which looks like a rung with one contact and coil with a dashed line beneath it.

The IEC-compliant timers are system function blocks and are added to the program blocks when the first one is placed in the program. To place a timer block in a network:

Position the cursor where you want the timer block to appear.

The blocks are shown in the pane on the left side of the window. Expand the "Libraries" entry and then expand the "Standard Library" folder. Expand the "System Function Blocks" folder. Note that the first six entries (SFB0 to SFB5) are the normal timer and counter blocks.

Double-click on the "SFB4 TON" line. A TON timer is added to the current rung.

Now, add a symbol for the timer. Start the symbol editor by selecting **Options | Symbol Table**. Note that the TON symbol with address and symbol type of SFB4 has been added.

Add a new timer symbol (e.g., "Tmr1" to the symbol column of the last line of the symbol list. The address should be "DB1" and the data type is "SFB4." Actually, the DB (data block) number can be any number as long it is not already assigned as a data block. The data type must match the data type of the TON timer (SFB4, in this case).

Save the new symbols by clicking on the disk icon in the menu bar.

Exit the symbol editor.

Now change the "???" above the TON block to the timer symbol you defined in the previous step. Press <Enter>. You will be asked if you want to generate the data block. Answer "Yes."

Place the logic for the IN timer input. You can add contact logic to the EN and IN inputs. Generally, the EN input is connected directly to the power rail.

To specify the preset (PT) input, click on the PT input. Type in the preset value in the IEC format for time.

You cannot connect a coil to the Q output. You can connect it to a symbol.

Since we added a SFB to the program, we must download it from the project manager window (that shows the program blocks). Save OB1 and then exit from the LAD/STL/FBD editor. Click on the download icon. You will need to answer "Yes" to the prompts about overwriting blocks. You do not need to load the system data.

Re-open OB1 and observe the program operation.

4. Counters

An up-counter is SFB0. The procedure to add an up-counter is very similar to that of adding a timer, except that the data type for the counter data block is "SFB0". To place a counter block in a network:

Position the cursor where you want the counter block to appear.

Double-click on the "SFB0 CTU" line in the "System Function Blocks" folder. A CTU counter is added to the current rung.

Now, add a symbol for the counter. Start the symbol editor by selecting **Options | Symbol Table**. Note that the CTU symbol with address and symbol type of SFB0 has been added.

Add a new counter symbol (e.g., "Cnt1" to the symbol column of the last line of the symbol list. The address should be "DB3" and the data type is "SFB0." Actually, the DB (data block) number can be any number as long it is not already assigned as a data block. The data type must match the data type of the CTU counter (SFB0, in this case).

Save the new symbols by clicking on the disk icon in the menu bar.

Exit the symbol editor.

Now change the "???" above the CTU block to the counter symbol you defined in the previous step. Press <Enter>. You will be asked if you want to generate the data block. Answer "Yes."

Place the logic for the CU and R counter inputs. You can add contact logic to the EN, CU, and R inputs. Generally, the EN input is connected directly to the power rail. To specify the preset (PV) input, click on the PV input. Type in the preset value as an integer.

You cannot connect a coil to the Q output. You can connect it to a symbol.

To display the CV while the program is executing, you must connect it to an INT variable.

Since we added a SFB to the program, we must download it from the project manager window (that shows the program blocks). Save OB1 and then exit from the LAD/STL/FBD editor. Click on the download icon. You will need to answer "Yes" to the prompts about overwriting blocks. You do not need to load the system data.

Re-open OB1 and observe the program operation.

C. ADDING COMMENTS TO LADDER PROGRAM

Comments are placed inside the Comment box that appears with each network.

D. ASSIGNING SYMBOLS TO ADDRESSES

To assign symbols to addresses, an organization block must be open. The default organization block is OB1. To open it, keep expanding the project "tree" in the left pane until the "Blocks" folder is visible. Select the "Blocks" folder. The OB1 block should now be visible in the right pane. Double-click on "OB1" to open it. To access the symbols, from the menu select **Options | Symbol Table**. To assign symbols to an address,

Click in the symbol column of the last row.

Type the name of your symbol and press <Tab>.

The address column will now contain the next address after the address in the previous row. The address may be highlighted in pink if it is the same as another address already existing in the data base. Change the address to what is needed and press <Tab>. The data type will change to a type consistent with the address. If you really want INT (for integer) or REAL (for real values), you will need to change the data type. The default type for a MDx address is DWORD.

Press <Tab> to place the cursor in the Comment column.

Type in a description and then press <Tab>.

Enter the other input symbols. Note that the address automatically increments to the next one.

To close the symbol editor, click on the red "X" in the upper right corner.

E. DISPLAYING MEMORY LOCATIONS

To display memory locations, an organization block must be open. The default organization block is OB1. To open it, keep expanding the project "tree" in the left pane until the "Blocks" folder is visible. Select the "Blocks" folder. The OB1 block should now be visible in the right pane. Double-click on "OB1" to open it. Click on the eyeglasses icon to go on-line.

While on-line, a window should appear at the bottom of the screen. Select the Modify tab. A table with one blank row should appear. To add the symbol column, right-mouse the header of the "Address" column, select "Display columns (on/off)" and then select "Symbol".

Type the name of the symbol you wish to display in the Display column. The value will appear in the "@ Status value" column. To change the format, right-mouse the "Display" column for that row, and select the appropriate format (Boolean for internal coils, decimal for integers, or floating point for reals).

To modify a particular location that is not an input/output channel, enter the new value in the "Modify value" column. To actually change the address to that value, right-mouse the cell with the new value, and select "Modify". For Booleans, it is not necessary to enter a value in the "Modify value" column. One can right-mouse any cell on the row and select "Modify address to 0" or "Modify address to 1".

F. INPUT/OUTPUT FORCING

For inputs, the force function allows one to override the actual status of external inputs by forcing individual discrete inputs to be ON or OFF. Input forces are written to the input image table and have no effect on the actual input. The value in the force table overrides the actual input. For example, if I0/1 is forced to 1 (ON), it will appear in the input image and ladder logic as 1, regardless of the actual status of the device.

For outputs, the force function allows one to override the processor logic and force individual discrete outputs to be ON or OFF. Output forces are written directly to the output device and do not appear in the output image table. The value in the force table overrides the value in the output image. For example, if Q4/2 is forced to 1 (ON), the actual output will be ON, regardless of how it appears in the ladder and output image table.

Note: Enabling or disabling forces can result in sudden machine movement, possibly injuring personnel. Applying or removing forces while forces are enabled takes place immediately, with no prompt for confirmation. **USE EXTREME CAUTION WHEN USING FORCES!**

Note: for the S7-300 PLC, discrete outputs cannot be forced.

To force discretes, the Force Values Display must be invoked. The individual inputs or outputs are forced from this display. To force values:

From the Step 7 manager menu, select **PLC | Display Force Values**. The Force Values window pops up.

If the processor status is not displayed at the bottom, connect it to the processor by selecting **PLC | Connect to | Configured CPU**.

Type the address or symbol of the discrete input to be forced in the table.

To force a discrete,

Type the force value ("1", "0", "true", or "false") in the Force Value column.

Right-mouse the force value and select **Force**.

When prompted if you want to continue, answer "Yes."

To remove a force

Right-mouse the force value and select **Remove Force**.

When prompted if you want to continue, answer "Yes."

The entry is deleted out of the table.

If a discrete is forced, a red "F" will appear in the first column of the Force Values display. A forced discrete in a monitored program will appear with a yellow highlight.

G. IMPORTING/EXPORTING SYMBOLS

1. Importing Database from DIF file

To import an address/symbol/description database from a .DIF (data interchange format) file:

First open the symbol table. Open an organization block (usually OB1). From the menu select **Options | Symbol Table**.

From the symbol table menu select **Symbol Table | Import**.

Select the appropriate file. Click on **Open**.

A pop-up will remind you that you cannot undo the action. Click on **Yes** to continue the import.

The symbols are imported.

A pop-up will appear with a summary of the errors (if any). If you click on **Yes**, a log of the import operation will be displayed. If there are no errors, you should click on **No**.

2. Exporting Database to DIF file

To export an address/symbol/description database to a .DIF (data interchange format) file:

First open the symbol table. Open an organization block (usually OB1). From the menu select **Options | Symbol Table**.

From the symbol table menu select **Symbol Table | Export**.

Select the appropriate file name. Click on **Save**.

The symbols are exported to the file.

A DIF file can be edited with Excel.

H. SAVING LADDER LOGIC PROGRAMS

Ladder logic programs (projects) may be saved for future use. By saving your ladder logic programs, you are able to resume your experiment, even though others have been using the same PLC. One usually saves the program to the C:\Projects\Siemens program directory on the hard disk and then saves the program to a flash drive for backup/transport to another computer.

From the SIMATIC Manager, select **File**

To save with a different project name, select **Save As...**

To save your project to another directory, use **Save As...** and select the proper drive and directory.

A more efficient way of storing programs is to archive them in a .zip file. A project has more than 150 files and more than 30 subdirectories. To archive a program,

From the SIMATIC Manager, select **File / Archive..**

Select your project and click on **OK**

Select the directory in which to place the archive and its name. The default name is the first 8 characters of the project name. Click on **Save**.

The files are zipped into the selected file.

DO NOT USE Winzip, or any of its variants to save your project. You must use the Archive feature of the SIMATIC manager. If you use Winzip, the project will be irretrievable and cannot be restored.

I. RESTORING LADDER LOGIC PROGRAMS

You may restore your ladder logic project by copying a previously saved project directory from a flash drive or your account or retrieve a previously archived project. After copying the project directory to the appropriate place (usually C:\Projects\Siemens\), select **File / Open** and then click on **Browse** and then select the project directory.

If you do not want to copy the project, but merely access it on your flash drive, select **File / Open** and then click on **Browse** and then select the project directory.

To restore a project archived in a .zip file,

From the SIMATIC Manager, select **File / Retrieve..**

Select your project and click on **OK**

Select the directory in which to place the project and click on **OK**.

A pop-up summarizing the operation is displayed. Click on **OK**.

Another pop-up will ask if you want to open the project. Click on **Yes** if you want to open the project.

J. PRINTING A LADDER LOGIC PROGRAM

To print program, for each program block (except for SFC, S7-Graph)

- Right-mouse block

- Select Print | Object Content

To print S7-Graph block

- Open source

- Select File | Print Preview

- Select “According to Settings” and click on “Settings” button

- On Views select Sequencer, Block Comment, Conditions and action, Symbolic representation: LAD

- Click OK on this window and the next.

- Make sure print preview is fine

- Click “Print” to print to Distiller

To print symbol table

- Double-click on OB1 block to open it

- Select Options | Symbol Table

- Click on printer icon

To print cross-reference

- Display project in SIMATIC Manager

- Select Options | Reference Data | Display

- Select Cross Reference

- Click on printer icon