

Siemens Step 7

S7-400

Programming

Table of Contents

	Page
A. Starting a Project With the S7-400	Step7_400-3
1. Configuring Communications	Step7_400-3
2. Rack Configuration	Step7_400-4
3. Specifying Other Variables	Step7_400-6
4. Processor Download	Step7_400-7
B. Ladder Logic Programming	Step7_400-8
1. Two Contacts in Series (Logical AND)	Step7_400-8
2. Entering Branches (Logical OR)	Step7_400-9
3. Timers	Step7_400-9
4. Counters	Step7_400-10
C. Adding Comments to Ladder Logic	Step7_400-11
D. Assigning Symbols to Addresses	Step7_400-11
E. Displaying Memory Locations	Step7_400-12
F. Input/Output Forcing	Step7_400-12
G. Importing/Exporting Symbols	Step7_400-13
H. Saving Ladder Logic Program	Step7_400-14
I. Restoring Ladder Logic Program	Step7_400-14
J. Printing Ladder Logic Program	Step7_400-15

Step7 - 2

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 6ES7 407-0KA01-0AA0 PS407 10A power supply

Siemens 6ES7 417-4XL00-0AB0 CPU417-4 (Rev 3.0) processor

Siemens 6ES7 6GK7 443-1EX10-0XE0 CP 443-1 (Rev 2.3) Ethernet comm. Module

Profibus network to an ET200M rack with the following modules:

Siemens 6ES7 153-2BA00-0XB0 IM 153-2 interface module

Siemens 6ES7 321-1BL00 32 chan. 24v disc. input

Siemens 6ES7 322-1BH01 16 chan. 24v disc. out/0.5A

This document is primarily for version 5.4 of the Step7 software. A **SERIAL COMM MODULE** serial cable connection from the PC to the processor is also assumed.

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 Step 7 version 5.4 is used to program the S7-300 and S7-400 PLCs. An icon for these programs should already be on the desktop.

The PLC and the PC used to program it are connected directly through a serial RS-232 link.

A. STARTING A PROJECT WITH THE S7-400

Wire the switch/light board before proceeding. Power up the computer, the PLC rack, and external power supply, as needed. Turn the key on the PLC processor to the RUN-P position.

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

Since the PC serial port is used to connect to the processor, the RSLinx AB_DF1 driver must be stopped before Step7 will be able to connect to the processor.

Start the Step 7 software by clicking on the SIMATIC STEP 7 icon on the desktop, or by selecting

Start | All Programs | SIMATIC | SIMATIC Manager

A project window should appear with a list of selections. If you are starting a new project and a project is displayed in the window, close it by clicking on the red "X". The easiest way to start a new project is to use the "New project" wizard:

Select from the program menu, **File | 'New Project' wizard...**

Click on **Next>**.

Select CPU417-4 as the CPU type. Click on **Next>**.

The OB1 block should be checked. Select "LAD" as the language for the OB1 block.

Click on **Next>**.

In the "Project Name" field, type in your project name. You should make the first 8 characters unique. Click on **Finish**.

The software will start up in the offline mode and a blank project is displayed. The actual project consists of a number of files contained in a directory. Note that the directory that contains the project is shown in the title bar of the project window.

Also note that it has been truncated to 8 characters.

To save the project to the C:\Projects\Siemens directory, select **File | Save As** and select the appropriate project storage location (C:\Projects\Siemens)

Click **OK**.

If your project is already in the PLC, click the "open file" icon and select your project from the list of projects, and then click on **OK**.

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.

1. Configuring Communications

Set up project communications. Right-mouse the name of your project in the left window pane, Select **Insert new object** and select **PG/PC**. Press enter to keep the default name. To set up serial port communications:

Right-mouse **PG/PC(I)** in the right window pane. Select **Open Object**.

Click on the "Interfaces" tab. Click on the **New** button.

Select "MPI" and click on **OK**.

Set the address to "3" and select (highlight) the MPI in the Subnet field. Click on **OK**.

Step7 - 4

Click on **OK**.

Now verify that the PG/PC interface parameters are correct.

From the main program menu, select **Options | Set PG/PC Interface...**

Select the "PC Adapter (MPI)" line.

Click on **Properties**.

Click on the "MPI" tab. Verify that "PG/PC is the only master on the bus" is checked and the Address is "3".

Click on the "Local Connection" tab. Verify that the COM port is "1" and the transmission rate is "19200". Click on **OK**.

Click on **OK**.

Insert **PROFIBUS** and **Industrial Ethernet** communications in the same manner as done for the PG/PC interface. There are no parameters to set.

You are now ready to configure the processor rack.

2. Rack Configuration

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

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

The second step is not absolutely necessary, but it is easier to specify symbols 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 variable.

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

Slot	Module	Order Number	Firmware
1, 2	PS407 10A	6ES7 407-0KA01-0AA0	
3, 4	CPU417-4	6ES7 417-4XL00-0AB0	V3.0
5	CP 443-1	6GK7 443-1EX10-0XE0	V2.3

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

Highlight the SIMATIC 400 Station in the left pane (If not displayed, click on the "+" by name of project on first line)

Double-click "Hardware" in the right pane.

The window displays a rack with the CPU already in slot 4 (and 5).

If a right pane, displaying a catalog of available items, is not available, select View | Catalog from the main menu.

To insert a module into the configuration, the module can be selected from the catalog in the right pane, or the slot can be right-clicked and the module selected

from the pop-window. The latter method is harder since the catalog number of the module is “flashed” as a hint. Therefore, the former method is described.

First insert the power supply. Expand the SIMATIC 400 category and then the PS-400 folder and then the Standard PS-400 folder. Select the appropriate module (PS 407 10A, the second one). Watch the catalog number in the lower window and verify that it matches the number in the table above.

Now drag-and-drop the module to slot 1 of the rack displayed on the left. It should appear in slot 1.

Unfortunately, the new project wizard assigns the processor to the wrong slots, so it must be moved to the correct slot. Since it is two slots wide, you cannot just move it to slot 3. Move it to slot 6 by left-clicking (and holding down the mouse button) on the slot 4 entry (first column) and drag it to slot 6. Now move the processor to slot 3.

The version number of the module must be changed to 3.0. To accomplish this, right-click on the row containing the CPU. Select Replace Object | CPU 417-4 | 6ES7 417-4XL00-0AB0 | V3.0 Click on “Yes”.

Now select the Ethernet communication module from the catalog in the CP-400 and Industrial Ethernet folders. Make sure you select the proper version. When the module is dragged to the rack, a popup window will appear in which you will need to specify the MAC and IP addresses.

Set the properties as follows:

MAC Address: MAC of module

IP Address: whatever is assigned by IT

Subnet mask: 255.255.255.255

Gateway:

Use Router (dot in button)

Address: whatever is assigned by IT

Highlight “Ethernet” in the Subnet window.

Click on “OK”

Now set up the PROFIBUS communication link to the I/O rack.

Right-mouse the row in the CPU module containing “X2”.

Select “Add Master System”.

Highlight the PROFIBUS row. Click on “OK”.

Select the interface module (IM153-2). It is in the PROFIBUS-DP catalog category ET200M folder. Select the catalog number in the table below. Drag the IM 153-2 to the PROFIBUS line in the upper left pane. A popup window will appear in which you will need to specify the address. It should be “3”.

The remote rack will be set up as follows:

Slot	Module	Description	Start Addr.
2	IM 153-2	6ES7 153-2BA00-0XB0	
4	6ES7 321-1BL00	32 chan. 24v disc. input	I0
5	6ES7 322-1BH01	16 chan. 24v disc. out/0.5A	Q4

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

Expand the IM 153-2 folder and then the DI-300 folder. Select the appropriate module (SM321 DI32x24V, the first one). Watch the catalog number in the lower window and verify that it matches the number in the table above.

Now drag-and-drop the module to slot 4 of the rack displayed on the lower left pane.

It should appear in slot 4 and the I addresses should be automatically assigned.

Expand the DO-300 folder. Select the appropriate module (SM322 DO16x24V/0.5A).

Watch the catalog number in the lower window and verify that it matches the number in the table above.

Now drag-and-drop the module to slot 5 of the rack displayed on the left. It should appear in slot 5 and the Q addresses should be automatically assigned.

Change the start address of the discrete output module by double-clicking on the row and change the start address to "4".

Click on the Save icon (floppy disk).

Close the hardware configuration window by clicking on the red "X."

To assign symbols to the input/output channels, an organization block must be open. The default organization block is OB1. To open it, keep expanding the CPU 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. A window with a blank network 1 should open. To access the symbols, from the menu select *Options* | *Symbol Table*. A symbol list with only OB1 defined will appear. To assign symbols to the discrete input channels,

Click in the symbol column of the second row.

Type "Start_PB" and press <Tab>.

The address column will now have "OB 2". Change the address to "I0.0" and press <Tab>. The data type will change to BOOL.

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

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

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

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

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

Close the OB1 editor window by clicking on the red "X" in the upper right corner.

Close OB1 (click on red "X" in upper right corner).

3. Specifying Other Variables

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

4. Processor Download

First verify that the PG/PC is set to download to the processor over Ethernet.

On menu bar at top of screen, click on "Options"

Select "Set PG/PC Interface"

Make sure "ISO Ind. Ethernet – Broadcom..." (without yellow triangle) is highlighted

Click on "OK"

To download to the processor,

Highlight "Blocks" in the left pane.

Click on the Download icon (yellow arrow pointing down to a PLC rack).

A popup window will ask if you want to overwrite objects. Unless you know what you are doing, you should answer "Yes" to this prompt.

A popup window will ask if you want to load system data. When downloading for the first time, answer "Yes" to this prompt. On subsequent downloads due to program changes, answer "No" to this prompt, unless you are changing system data (e.g., error log).

If you answered "Yes" to the last prompt, a popup window will ask if you want to delete the system data. Answer "Yes" to this prompt.

You may get a prompt about the system data not being current and asking if you want to perform a download. Answer "Yes" to this prompt. The next window will show the differences. If the only difference is the module name, click on "OK". If there are other differences (wrong module # or firmware version), click on "Cancel" and correct these items in the hardware configuration before attempting a download.

If a prompt appears indicating that the processor will be stopped, answer "OK."

If a prompt appears asking whether you want to start the module, answer "Yes."

Go to the next section to start 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 discretely, 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.

K. 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