ARL-TN-1309 ● MAY 2026

Programming the MY700 Jet Printer with FactoryLogix

by Kenneth Pugh

NOTICES

Disclaimers

The findings in this report are not to be construed as an official Department of the Army position unless so designated by other authorized documents.

Citation of manufacturer's or trade names does not constitute an official endorsement or approval of the use thereof.

Destroy this report when it is no longer needed. Do not return it to the originator.

ARL-TN-1309 ● MAY 2026

Programming the MY700 Jet Printer with FactoryLogix

Kenneth Pugh
DEVCOM Army Research Laboratory

REPORT DOCUMENTATION PAGE

1. REPORT DATE	2. REPORT TYPE	3. DATES COVERED	
		START DATE	END DATE
May 2026	Technical Note	February 12, 2026	March 26, 2026

4. TITLE AND SUBTITLE
Programming the MY700 Jet Printer with FactoryLogix

5a. CONTRACT NUMBER	5b. GRANT NUMBER	5c. PROGRAM ELEMENT NUMBER
5d. PROJECT NUMBER	**5e. TASK NUMBER**	**5f. WORK UNIT NUMBER**

6. AUTHOR(S)
Kenneth Pugh

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)	8. PERFORMING ORGANIZATION REPORT NUMBER
DEVCOM Army Research Laboratory ATTN: FCDD-RLA-WE Aberdeen Proving Ground, MD 21005	ARL-TN-1309

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)	10. SPONSOR/MONITOR'S ACRONYM(S)	11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT
DISTRIBUTION STATEMENT A. Approved for public release: distribution unlimited.

13. SUPPLEMENTARY NOTES
ORCID: Kenneth Pugh, 0009-0003-8430-1467

14. ABSTRACT
ARL developed a streamlined process for programming the MY700 printed circuit board (PCB) solder jet printer using FactoryLogix software to support in-house PCB fabrication for custom diagnostic sensor suites. The MY700 jet printer—a high-speed dual-jet dispensing platform—enables precise solder paste application, reducing manual labor while improving accuracy and efficiency during PCB assembly. This report outlines the step-by-step process for creating a new assembly program using Altium Designer for PCB design, FactoryLogix for program generation, and a jet printing interface for verification and customization. The report also details the setup of PCB panels, fiducial assignment, component selection, and solder paste deposit configuration, ensuring reliable program transfer to the MY700 system. Furthermore, the report addresses network restrictions that require stand-alone systems and approved file transfer methods. Key sections include electronic computer-aided design for PCB layout adjustments, part assignments for solder paste application, and line programming for program generation and transfer. Common issues and troubleshooting tips are provided to ensure consistent and error-free operation. This report serves as a comprehensive guide for ARL's Weapon Guidance and Fuzing Branch, enabling efficient and accurate PCB manufacturing practices tailored to their specific equipment and software configurations.

15. SUBJECT TERMS
Weapons Sciences, MY700 Jet Printer, FactoryLogix Programming, Transfer Tool, PCB assembly, Jet Printing Interface, JPI

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES
a. REPORT	b. ABSTRACT	C. THIS PAGE		
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	SAR	17

19a. NAME OF RESPONSIBLE PERSON	19b. PHONE NUMBER *(Include area code)*
Kenneth Pugh	(520) 691-5306

STANDARD FORM 298 (REV. 5/2020)
Prescribed by ANSI Std. Z39.18

Contents

List of Figures

1. Introduction

The MY700 is a high-speed dual-jet dispensing platform developed by Mycronic.[1] It can dispense a wide range of materials, including solder paste, at rates of up to 300 dots per second. The system applies precision solder paste deposits that aid in the production of accurate and reliable solder joints on printed circuit boards (PCBs) after component placement and reflow.[2] When paired with FactoryLogix software, the MY700 provides flexible programming capabilities that make it well-suited for custom PCB manufacturing.

The U.S. Army Combat Capabilities Development Command (DEVCOM) Army Research Laboratory (ARL) designs and builds custom embedded diagnostic sensor suites for various projectile experiments to obtain aerodynamic flight data. The extensive customization of these sensor suites requires in-house PCB fabrication. To reduce build time and cost, ARL invested in the MY700 PCB solder jet printer (Figure 1). This system reduces manual labor while increasing speed and accuracy during PCB assembly.

Figure 1. Mycronic MY700 jet printer.

This report documents the steps needed to build a new assembly using the FactoryLogix software, which is used to program the MY700 printer.[3] This process is specific to the equipment, software configuration, and manufacturing practices of ARL's Weapon Guidance and Fuzing Branch. Adhering to this procedure helps ensure consistent setup, correct component assignment, and reliable program transfer.

2. Description

The MY700 is used to populate and manufacture custom PCBs designed in-house by engineers and technicians. The MY700 accurately applies solder paste to designated component locations before placement and reflow. The following tools and processes are needed to successfully operate the MY700:

- Altium Designer Software: Altium is used to design the PCBs developed at ARL. This software creates electrical schematics, performs PCB layout, and generates manufacturing files (ODB++) required to program the MY700.
- FactoryLogix Software: FactoryLogix uses the ODB++ files generated from Altium Designer to create a PCB assembly program for the MY700.
- Jet Printing Interface (JPI): The JPI is an extension of FactoryLogix. Inside of JPI is Jet Printing PCB Data Editor, which is used is to verify that all fiducials, shapes, and components are properly assigned before sending the program to the Transfer Tool. Obstacle height settings can be adjusted here, along with solder deposits, such as dot size and dot shape. This is also where the program is labeled as either Top or Bottom and generated to the Transfer Tool.
- Transfer Tool: The Transfer Tool transfers the generated program from FactoryLogix/JPI to the MY700 system.
- CD-R disk or other approved file transfer method.
- Bare PCB panel for the current project.
- Ruler and scale (showing mils) used to measure board frame and fiducial locations.
- PDF Documentation: This is generated by Altium Designer, which includes electrical schematics and PCB layout images.

Due to network restrictions at ARL, the MY700, FactoryLogix software, and Transfer Tool are stand-alone systems and not part of the network, whereas the Altium software is on the network. Therefore, direct transfer of the ODB++ files is not possible. To transfer the files, an approved transfer method (i.e., CD-R) is required.

3. Assembly Project Setup: Getting Started

This section outlines the steps needed to generate the paste program required for a PCB panel. This version of the program includes using ODB++ files to program MY700.

The steps for FactoryLogix programming are as follows:

1) Verify that the MY700 system is powered ON. On the stand-alone PC, open the FactoryLogix program and select Login. (No password is required.)
2) Proceed to **Process Engineering** and select **New Assembly**. This will open the **New Assembly Information** screen. In this section, the user must enter the new assembly name and revision level.
3) Below the Assembly Revision field, there are two options for creating the assembly: **New BOM** or **Copy from Existing Assembly**. Select **New BOM** when creating a completely new project. Select **Copy from Existing Assembly** when you want to copy an existing assembly from the FactoryLogix assembly folder for new revision.
4) Enter the new assembly information, select the appropriate assembly option, and click **Next**. This will take you to the **Load Design Files** page. The Load Design Files option is used to import the ODB++ project file.
5) At the top of the **Load Design Files** screen, click **Add File**.
6) Navigate to the ODB++ folder that was previously copied and select the file named **matrix** within that folder.
7) Click **Open** to load the ODB++ matrix file into the system.
8) After the file has been successfully loaded, click **Next** to proceed to the **Set CAD Options** screen. **Set CAD Options** allows the user to import additional information, such as electrical traces and part numbering. No selections are required in this section.
9) Click **Next** then select **Import**. After completing the import process, an image of the PCB will appear on the screen.

Note: If **Copy from Existing Assembly** was selected, then the **Add File** option is not required. Click **Next**, and the Assembly Library containing previously created assemblies will appear. Select the desired assembly from the list and click **Finish** to proceed.

4. PCB and Panel Setup

FactoryLogix software is organized into three primary sections: ECAD, Parts Assignments, and Line Programming. Each section serves a specific function in preparing and generating the program for the MY700 system. Brief descriptions of each section are provided below to explain their purpose within the PCB and panel setup process.

4.1 ECAD

During the PCB and panel setup stage, the ECAD section is used to review and configure the layout of the PCB before programming the MY700. Selecting the

ECAD option displays the PCB image and provides access to several tabs that allow adjustment of board features, creation of fiducials, and configuration of panel and frame settings. Depending on the original PCB design, the board frame and panel layout may already be imported. They can be created or modified in ECAD if needed.

One critical function in this stage is the creation and assigning of fiducials. Normally 0.5 to 1.0 mm in diameter, fiducials are small, circular points the board designer adds to the corner of the PCB design to ensure precise alignment and orientation with the MY700. When assigning fiducials in FactoryLogix, the user selects either **Panel fiducials** for single board builds or **Circuit fiducials** when working with multiple boards in a panel. Once selected, fiducial properties can be reviewed and adjusted before finalizing. Proper fiducial setup ensures accurate board alignment and proper paste deposits.

The **Frames** tab is used to create or modify the panel frame if one is not already defined. The panel frame creates the board dimensions and is used to determine conveyor width settings on the MY700. Users can manually enter height and width dimensions or create the frame from rail-to-rail parameters. Accurate framing ensures proper board insertion into the MY700.

If multiple PCBs are fabricated in a single panel, the **Panel Layout** tab is used to add Panelization. Using the **Create Matrix** function, the user defines the number of boards in the X and Y directions, along with spacing or offset between boards. In FactoryLogix, units can be specified in inches or millimeters.

These ECAD functions ensure that the PCB layout, alignment references, frame dimensions, and panel configuration are properly established before proceeding to the Parts Assignments section and program generation.

4.2 Parts Assignments

The Parts Assignments section is used to designate all component locations that will receive solder paste. Begin by confirming that the correct side of the board is selected. In the upper-right corner of the screen, under **Assigned Parts**, choose the appropriate board side before selecting any component locations.

Using the mouse, select the component locations for the assembly by positioning the cursor at one end of the desired area, holding the left mouse button, and dragging a selection frame around the required components. To select or deselect multiple individual components, use CTRL + Left Click. Do not include board fiducials in the selection. In addition, do not select non-component locations such

as mounting holes, edge connectors, or test pins unless they are specifically intended to receive solder paste.

After all required component locations have been selected, click the green arrow icon located at the top right of the screen. This action assigns the selected components locations for solder paste application. To confirm the assignment, click on an empty area of the board; properly assigned components will appear highlighted in yellow.

4.3 Line Programming

If programming a double-sided PCB, two separate assemblies are required: one for the top and one for the bottom side of the PCB. In FactoryLogix, only one side of the PCB can be assembled and generated at a time, so it is essential to select the correct side before proceeding.

The Line Programming section is where the board side is selected for generation. Under Line Programming, there will be options to choose Jet Printing Interface TOP Side or BOTTOM Side, depending on which side is being programmed. Select the appropriate option. In addition, verify that the TOP or BOTTOM icon located in the upper-right corner of the screen is correctly selected. This ensures that the proper side of the PCB is generated and prevents programming errors.

This section also includes additional tabs that support verification and setup. The **Show All Fiducials** tab allows you to confirm that all required fiducials are present and properly assigned prior to generation. There is also a **Rotation** tab, which allows the PCB orientation to be adjusted if required.

After selecting the correct board side, choose **Part Management & Troubleshooting**, then click **Next**, followed by **Setup**. This will open the **Jet Printing PCB Data Editor**.

5. Jet Printing PCB Data Editor

The Jet Printing PCB Data Editor has four main sections: Panels, Shapes and Components, Inspection, and Compiler. Each section is described below.

5.1 Panels Tab

Inside of Panels, the Assigned JPGroup is selected. The JPGroup is a configuration that defines which cassette models are installed on the MY700 system. Each cassette model is designed to hold specific types of solder paste or epoxy. Multiple JPGroups may be available depending on process requirements.

For standard assemblies, select the JPGroup named **Leaded**, unless otherwise specified by the process documentation or engineering.

Imported fiducials will also be visible within the Panels section. These should be verified to ensure proper alignment.

Another option within Panels is **Default Obstacle Height**. This sets the clearance height of the solder paste injector. The default value is 0. However, this value may need to be increased when fabricating boards with preinstalled tall components. Adjust the obstacle height as necessary to prevent interference during printing.

The final setting within the Panels section is the **Conveyor Clamp Type**. This option determines how the PCB is secured during processing. The default setting is Auto. If required, this may be changed to Light Grip to reduce board stress or other board gripping issues.

5.2 Shapes and Components Tab

Within the **Shapes and Components** section, there is a tab labeled **Purpose**. A Purpose is a predefined configuration that determines the solder or epoxy type, deposit volume, and associated cassette model.

The standard Purpose is **Leaded**, which is preconfigured and should be used for standard assemblies unless otherwise specified. Additional Purposes may be created for specific PCB assemblies.

A required step within both Shapes and Components is assigning each shape and component to the selected Purpose. This ensures the correct solder or epoxy material and volume are applied during printing.

To automatically assign the selected Purpose:

1) Click the blue **Auto Match** icon.
2) Both Shapes and Components will be flagged with the **Leaded** Purpose.

To unassign a specific shape or component:

1) Select the desired shape or component.
2) Click **Flags**.
3) Select **D (No Deposits)**.

This action updates both Shapes and Components simultaneously.

Deposit Editor is also available within both the Shapes and Components sections. This tool allows you to modify the volume, shape, and size of solder or epoxy deposits. Editing deposits under **Shapes** applies changes to all selected shapes

within the assembly. Editing deposits under **Components** allows adjustments to individual components only. Use the Deposit Editor when process adjustments are required.

5.3 Inspection Tab

The **Inspection** tab provides the option to inspect either all components or selected components within the assembly. During inspection, the MY700 system evaluates solder deposits and detects missing or deformed deposits. By default, all components are selected for inspection. To exclude a specific component:

1) Select the corresponding component from the list.
2) Click the green icon next to **Flags** to toggle inspection off.

This will remove the selected component from the inspection process while leaving all other components enabled.

5.4 Compiler Tab

The **Compiler** tab is where the assembly program is generated and sent to the Transfer Tool. During this step, the assembly must be named according to the board side being processed. Use **Top** when generating the top side of the board. Use **Bot** or **Bottom** when generating the bottom side of the board. This designation will be added to the initial name created during the **New Assembly** setup.

Click **Generate Program**. The completed assembly will then be transferred to the Transfer Tool.

6. Transfer Tool

The **Transfer Tool** is used to send the newly generated assembly to the MY700 system. On the **FactoryLogix PC**, locate and open the **Transfer Tool** from the desktop. Once opened:

1) Select the newly generated assembly.
2) In the upper-right corner of the screen, click **Transfer**.

The new assembly will then be transmitted to the MY700 system library.

7. Notes and Common Issues

- Verify the correct Top or Bottom board side is selected before assigning parts and generating the program.

- Ensure all fiducials are properly imported and assigned in the **Panels** tab of the JPI prior to compiling.
- Confirm that all required components are assigned **User Leaded** purpose and that non-soldered items are skipped.
- The MY700 must be powered on before using the **Transfer Tool**, or the program transfer will fail.
- If the program does not appear in the **MY700** folder after transfer, rerun the **Compiler** step and repeat the transfer process.
- Review any flags in the Shapes tab and resolve them using Auto Match before proceeding.
- A common error message when using FactoryLogix is “Cannot import because there are no fiducials.” This error can be resolved by resetting FactoryLogix.

8. References

1. Mycronic. MYPro series MY700 jet printer and dispenser specification; c2025 [accessed 2026 Feb 12]. https://www.mycronic.com/globalassets/global-blocks/product-areas/pcb-assembly/_pdf/ab59823-my700-jp-jd-jx-specification-flyer-march-2022_lr.pdf
2. Mycronic. AB59821 MY700 brochure; 2021 Sep [accessed 2026 Feb 12]. https://www.mycronic.com/globalassets/global-blocks/product-areas/pcb-assembly/_pdf/ab59821-my700-brochure-september-2021_lr.pdf
3. Aegis Industrial Software Corporation. AEGIS software, Factory 4.0. c2026 [accessed 2026 Feb 12]. https://www.aiscorp.com/factory-4-0/industry-4-0/

List of Symbols, Abbreviations, and Acronyms

ARL	Army Research Laboratory
CAD	Computer-Aided Design
DEVCOM	U.S. Army Combat Capabilities Development Command
ECAD	Electronic Computer-Aided Design
JPI	Jet Printing Interface
PC	Personal Computer
PCB	Printed Circuit Board

1 (PDF)	DEFENSE TECHNICAL INFORMATION CTR DTIC OCA
1 (PDF)	DEVCOM ARL FCDD RLB CB TECH LIB