

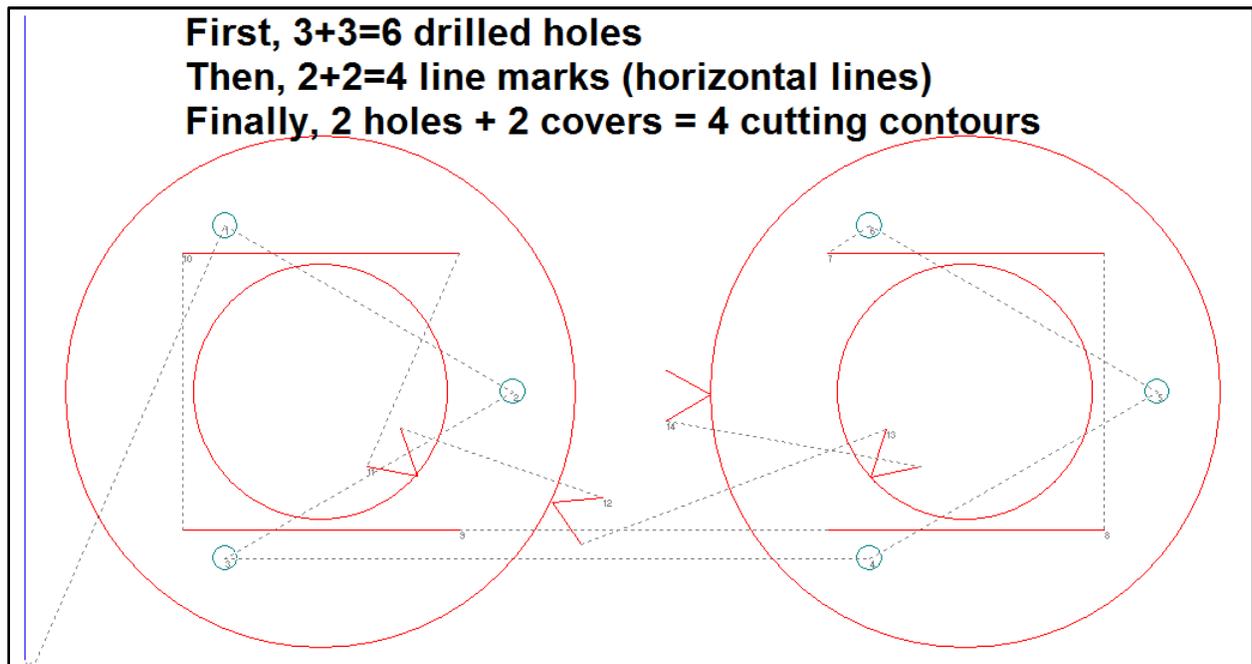
G2FABSOF

Date: 01.02.2021

Linemark & Drill – General Procedure

Introduction

There are number of ways to combine drilling, linemarking and cutting cycles in a cnc code file. One such coding style is shown below using 2 similar nested parts, each of which contains 2 drilled holes, 1 marking line, 1 hole and 1 cover profile. (As indicated in the picture below, drilled holes can also be used as punchmarks. In general, the software names any machining process that is completed at a single point as **drill** and any geometric contour that is drawn on the surface of a nested part as **linemark**. This naming convention will be followed in the rest of the document.)



G2FABSOF

Date: 01.02.2021

Drawing

It is necessary to identify each type of profile (drill, linemark, cut) correctly within the part drawing file. You can do this in many ways (like, keeping the profile in an appropriate layer in a dxf file), as described in the software manual.

If you are new to the software, use the following method which is supported in our cad module (**II-D**), after converting a part drawing in our cad file format (**.mdf**).

1. Drill

- a. Make sure to draw (or, import) a circle of **diameter 5 mm** around the centre, wherever you would like to perform drilling operation.
- b. Press the **ESCAPE** key to un-select all the profiles.
- c. Select all the drill circles using mouse.
- d. Select the menu **Modify > Properties**.
- e. Click on the list appearing near **Type** and select **Drill**. (See the picture below.)
- f. Close the property window.

2. Linemark

- a. Make sure to draw (or, import) an open contour consisting of a series of lines & arcs, wherever you would like to perform linemarking operation.
- b. Press the **ESCAPE** key to un-select all the profiles.
- c. Select all the linemarking contours using mouse.
- d. Select the menu **Modify > Properties**.
- e. Click on the list appearing near **Type** and select **Linemark**.
- f. Close the property window.

3. Cut

- a. In general, all the closed contours in the part drawing are automatically identified as cover and holes. Modify their types, if necessary.

4. Check

- a. You can identify each profile type using a separate colour.
- b. Also, there is a separate menu for validating the drawing.
- c. Check the part drawing and apply necessary corrections, if required.

5. Save

- a. Save the modified part drawing in our cad file format (**.mdf**) using the menu **File > Save**. (The software may ask you to specify sequence as well as leadin/leadout while saving the part drawing file. Ignore by selecting **No**, in such cases.)

G2FABSOF

Date: 01.02.2021

Attributes ✕

Type:

Enclosure: for close profiles

Tool position: for open profiles

Class: 0 = Normal profile. Else, special profile.

Sequence no.: 0 indicates auto sequencing

Nestable hole Sub-class:

 0 - None

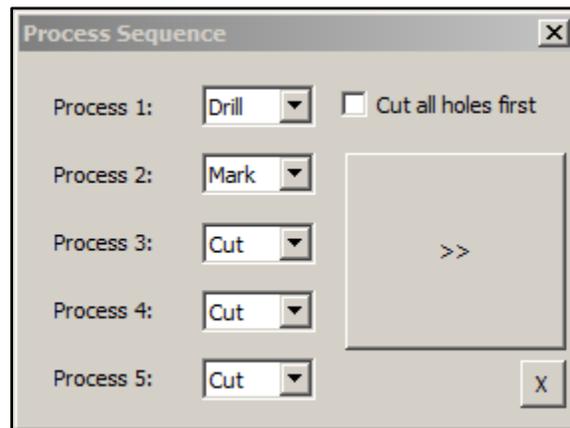
G2FABSOFT

Date: 01.02.2021

Sequencing

Nesting module works a bit differently, in this case. Irrespective of the sequence numbers displayed on the screen, the post-processor attempts to complete all the drilling operations in the entire layout, before generating codes for any linemarking or cutting process. That is why, it is desirable to generate the cutting sequence using the following procedure.

1. Generate the cutting sequence automatically, as usual.
2. Specify the process priority using the menu **Express > Process Priority**, as shown below. This will sequence all the drilling processes before any linemarking or cutting process.



3. Improve the sequence of linemarks and cutting cycles using following methods. (Note, all the drilling cycles should remain at the start.)
 - a. Semi-automatic methods using the following menus.
 - i. **Sequence > Sequence++ > Order cycles @ Corners**
 - ii. **Sequence > Sequence++ > Order cycles @ Touch**
 - iii. **Sequence > Sequence++ > Order pieces @ Corners**
 - iv. **Sequence > Order pieces @ Touch**
 - b. Manual method using the menu **Sequence > Manual Sequence**.
 - c. These methods mentioned above are described in a chapter named **Sequence** in the manual of the software.
4. At times, it may help to reverse the trace of a linemark using the menu **Cut > Reverse Sense**. This is because a linemark is an open contour and its start point may be far away from its end point, in certain cases.

G2FABSFT

Date: 01.02.2021

Settings

By default, the nesting module ignores all the cycles whose length is shorter than 10 mm. However, we have defined a drill as a circle with diameter **5 mm** and we do not want to ignore that. So, we have to modify the corresponding option, as described below.

1. Select the menu **Tools > Options**.
2. Select the tab **Toolpath** from the top of the window that appears on your screen.
3. Type **1** in front of **Minimum length > Cutting Cycle**. (See the picture below.)
4. Click on the **OK** button to close the window.

