Contents

CHAPTER 1: Getting Started

1.1 Main screen
1.2 View Modes
1.3 Using the mouse and the keyboard
1.4 Units
1.5 StartPoint and EndPoint
1.6 Coordinates

CHAPTER 2: OBJECTS

2.1. The block
2.2. Bulges
   2.2.1 Rectangular bulges
   2.2.2 Cylindrical bulges
   2.2.3 Polygonal bulges
   2.2.4 Arbitrary (odd) shaped bulges.
   2.2.5 Compose and Drawing inaccuracies
   2.2.6 Layers
   2.2.7 Odd shaped bulge editing
2.3. Pockets
   2.3.1 Cylindrical pockets
   2.3.2 Rectangular pockets
   2.3.3 Odd shaped pockets
2.4 Drills
   2.4.1 Composing a Drill
2.5 Threads
   2.5.1 Composing a Thread
   2.5.2 Thread type definition file
2.6 Longholes
2.7 Engravings
   2.7.1 Line Engraving
   2.7.2 Arc Engraving
   2.7.3 Circle Engraving
   2.7.4 Text Engraving
2.8 Free drawing
2.9 Fixtures

CHAPTER 3: MILLING TOOLS AND MATERIALS

3.1 Material Groups and Subgroups
3.2 Materials
3.3 Tool types
3.4 Importing older version Tool Data Base
CHAPTER 4: PROJECT SETUP

4.1 Defining the project filename
4.2 Defining the G-Code Program Number and filename
4.3 Defining the Block
4.4 Defining the end part
4.5 Attaching tools on the toolholder
   4.5.1 Automatic tool assignment
   4.5.2 Manual Tool assignment
   4.5.3 Selecting individual milling parameters

CHAPTER 5: COMPILATION

5.1 Inspecting the G-code
5.2 Make Options
5.3 Other options: chamfering

CHAPTER 6: ANIMATION

6.1 Running the compiled code
6.2 Running your own code

CHAPTER 7: THE TOOLBARS

APPENDIX I: GSIMPLE COMMANDS
CHAPTER 1: Getting Started

1.1 Main screen

The main screen of GSIMPLE includes the “Drawing and Animation Area”, the “Main menu”, the “Top Tool Bar”, the “Side Tool Bar” and the “Status Line”.

Below you can see a “normal” screenshot of GSIMPLE. The gray rectangle on the “Drawing and Animation Area” represents the machine travel limits. The red lines represent the work coordinates.

The “Top Tool Bar” contains buttons for the most common tasks such as opening and saving files, Project Setup, Object Definition (such as Pockets, Drills and Engravings), Project Compilation and Animation.

The “Side Tool Bar” is used for simple drawing (lines, arcs, circles etc), object modification (delete, move, turn etc) and changing between the different “View Modes”.

The “Status Line” contains information about the current mouse position and the current object (the last selected object).
1.2 View Modes

There are four "view modes" in Gsimple: Top, Axonometric, Isometric South East and Isometric South West. Drawing is possible only in Top View Mode. The View Mode may be selected through the View Menu Selection, the respective Side Tool Bar Buttons or the Right Mouse Button Popup Menu.

What is and what is not visible on the main screen is controlled through the View Options Dialog accessible through the View/View Options main menu selection.
1.3 Using the mouse and the keyboard

<table>
<thead>
<tr>
<th><strong>In order to...</strong></th>
<th><strong>Do What</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>Zoom in</td>
<td>Rotate the mouse wheel up</td>
</tr>
<tr>
<td>Zoom out</td>
<td>Rotate the mouse wheel down</td>
</tr>
<tr>
<td>Scroll screen (pan)</td>
<td>Move the mouse with the mouse wheel (or middle button) pressed down</td>
</tr>
<tr>
<td>To select a single object</td>
<td>Left click anywhere on it. All previous selected objects will be unselected. Depending on its type, the selected object will be colored red or blue. Red indicates that the object has been selected “as parent”. A parent is an object on which other objects may be attached. These terms are explained in detail in the object section.</td>
</tr>
<tr>
<td>To select multiple objects</td>
<td>Left click anywhere on it with the SHIFT key pressed down. The selected object will be colored blue. Previously selected objects will not be affected.</td>
</tr>
<tr>
<td>To unselect an object</td>
<td>Left click anywhere on it with the CTRL key pressed down.</td>
</tr>
<tr>
<td>To edit an object</td>
<td>Double click anywhere on it</td>
</tr>
</tbody>
</table>

Invoke the Right Mouse Button Popup Menu - Top View

- Start Point
- End Point
- Clear Points
- Snap Settings
- User Coord.
- Select
- Auto select
- Unselect
- Parent
- Cancel

Invoke the Right Mouse Button Popup Menu - 3D views (Isometric SE, Isometric SW, Axonometric)

- Top View
- Isometric SW
- Isometric SE
- Axonometric
- Zoom In (+)
- Zoom Out (-)
- View Options
- Cancel

Large and Small version of the Right Mouse Button Popup Menu
In order to... | Press
---|---
Edit selected object | Enter. If more than one objects have been selected Gsimple will respond with an error message.
Delete selected objects | Del. A list with all selected objects will appear and the system will ask the user for confirmation.

1.4 Units

Older Gsimple versions (prior to 2.04) used internally always the metric system. The user was able to choose the units he or she wished -but they were just used for entry and drawing. This resulted in constant recalculations -and precision loss!

Starting with version 2.04 a "UserUnit" has been introduced. This means the user must choose the unit system at installation time. The first time you use gsimple, you will be presented with the following dialog

The Company and User name fields are optional. Gsimple will use these data only in the “About” dialog box and “report printing”.

The data are stored in a file called gslogin.cfg This is also a simple text file, which you can edit through notepad. Alternatively, if you want to change these data you can run gslogin.exe (one of the executables supplied with gsimple).
1.5 StartPoint and EndPoint

Through the right-mouse-button menu you can set the Start and the End points. The StartPoint is marked by a small filled rectangular red spot. The EndPoint by a yellow one. The Start and End points are very useful for quick and accurate dialog data entry. For example if you set the start and endpoint at two arbitrary positions and then open the Draw Line dialog the From X,Y and To X,Y editboxes will be filled with the start and endpoint coordinates.

Picture: DrawLine dialog with the Start and End-points arbitrary set in advance

1.6 Coordinates

Gsimple has three sets of coordinates: machine coordinates, work coordinates and user coordinates.

The **machine coordinates** origin is commonly placed at the upper right corner of the machine limits. In GSIMPLE machine coordinates are fixed and do not affect in any way the drawing or the machine code.

**Work coordinates** are always defined in relationship to the machine coordinates origin. The user may select one of the 9 available in the standard g-code work coordinates (G54, G55, G56, G57, G58, G59.1, G59.2, G59.3). The selected coordinate system must correspond to the one selected in the machine itself.

The position of the work coordinates origin is defined through the following dialog (**Setup/Coordinates**)

It is a good practice to set X, Y approximately at the same values as it is set on the machine itself. This will enable you to see how your part is placed in the machine working envelop. The machine code positions are always relative to the work coordinates origin, so they are not affected by these values.
The “At block” checkbox will move your part and drawings placing the block left, lower corner at the work coordinates origin.

User coordinates are always defined in relationship to the work coordinates origin. All data entered during drawing are in “user coordinates”. The position of the user coordinates is defined through the following dialog (Setup/User Coordinates)

X,Y,Z are by default 0 - which means that the User and Work coordinates are by default identical. User coordinates do not affect the g-code. Normally you will set the User Coordinates origin at the lower, left corner or the center of your part.

You may also set the user X,Y coordinates at any point through the right-mouse menu.
CHAPTER 2: OBJECTS

There are 9 object types in Gsimple: The Block, Bulges, Pockets, Drills, Threads, Longholes, Engravings, Simple Drawings and Fixtures.

2.1. The block

The block (stock material) is a parallelepiped with its sides always parallel to the machine axis. The block cannot be turned or deleted. The block is normally the first object to be defined in a project.

The block is defined through the “Block dialog” (Setup/Block main menu selection)

Valid (positiv) data must be entered for the three block dimensions (length, width and height). Initially the block is placed at the coordinates origin, but may be moved wherever the user wants through the move command.

A block material must be selected from the supplied list. Gsimple uses the block material at compilation time in order to select the proper tool speeds, feeds and cutting depths from the tool-library.

The last field, labeled “Top” in the block dialog is determining the position of the block top face in accordance to the tool-zeroing-level.

A common practice in cnc-machining is to “zero” the tools on the topface of the block. In this case “Top” will be set equal to 0. In same cases the tools will be zeroed on the table, the vice etc. In this cases you must provide an accurate measurment of the distance between this level and the top face of the block.

**WARNING**

Setting “Top” (the distance between the block topface and the tool zeroing level) wrongly may result in a severe tool crash!

The block (if it is not selected) will be colored in green. If it is selected (not “as parent”) it will be colored in blue. If it is selected “as parent” in will be colored in red. You can hide the block, if you wish so, through the View/View Options dialog.
2.2. Bulges

A bulge is the opposite of a pocket. A pocket is an object with negative volume. A bulge has always positive volume.

A bulge must always reside on a “parent object”. Since the current version (2.04) does not accept “islands” (a bulge in a pocket) a bulge is allowed to reside only on the block or on another bulge.

In order to be able to draw a bulge you must have already selected its parent object “as parent”.

In order to facilitate the user GSIMPLE includes dialogs for the three most common bulge types - rectangular, cylindrical and polygonal. Odd (arbitrary) shaped bulges can be “composed” out of any closed-line drawing.
2.2.1 Rectangular bulges

Defining a rectangular bulge is done through the following dialog:

The dialog is divided into three parts: dimensions are entered in the upper left part, milling quality and tolerance in the upper right part and tools to be used and milling parameters in the lower part. The left, lower edge is assumed to be at the intersection of the left and the lower edge and X,Y are the distances from the user coordinate origin. Surface quality, chamfer, tolerance and Tools will be discussed later on.

If you try to open the rectangular bulge dialog (or any other dialog defining a bulge) with no object selected "as parent" Gsimple will complain with the following message:

2.2.2 Cylindrical bulges

The cylindrical bulge dialog is almost identical to the rectangular bulge dialog.

2.2.3 Polygonal bulges

The polygonal bulge dialog is almost identical to the rectangular bulge dialog. "Radius" is the radius of a circumscribed circle.
2.2.4 Arbitrary (odd) shaped bulges.

Arbitrary shaped bulges may only be "composed" from a drawing, through the following steps:

A. Draw the bulge outline using the “Draw line”, “Draw arc”, “Draw Circle” commands. Alternatively you can import a drawing from a DXF file, or use one of the existing “ready-made-drawing” dialogs (for rectangles, polygons and ellipses) or any combination of them.

B. Select the "parent object" by left clicking on it.

C. Select the lines and arcs that define the bulge by clicking on them with the shift key pressed in a **counterclockwise order**.

D. Open the **Compose Bulge Dialog** through the **Compose/Compose Bulge menu** selection or the Compose Bulge Top Toolbar button.

![Compose Bulge Dialog](image)

E. Type-in the height of the bulge, select tools (or leave the selection to Gsimple by selecting AUTO SELECT), select the finishing quality etc and click on **OK**.

Bulges are colored in white (if not selected). If they are selected "as parent" they are colored in **red**. If they are selected, but not "as parent", they are colored in **blue**.

Gsimple Version 2.04 has a restriction in the maximum number of elements (arcs and lines) a bulge may have and a restriction in the maximum number of bulges it can handle.

<table>
<thead>
<tr>
<th>Maximum number of lines and arcs in a bulge</th>
<th>128</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum number of bulges</td>
<td>32</td>
</tr>
</tbody>
</table>

www.gsime.eu
2.2.5 Compose and Drawing inaccuracies

Small drawing inaccuracies may prevent the conversion of a drawing into a bulge. In this case Gsimple will respond to the “compose bulge” command with a message like the following:

```
Can’t compose bulge. Shape is open!

⚠️ No connection between selected elements ID 3 and ID 4
```

This message means that the end point of the line or arc with ID3 does not coincide with the starting point of line or arc with ID4.

Gsimple does not demand a precise coincidence of these points. It can handle small deviations between them. In this case the original points will be substituted with a new one placed in the middle of them. The maximum allowed deviation is controlled by the “CON_TOLERANCE” configuration variable and may be
changed through the “Connection tolerance” dialog (accessed through Edit/Connection Tolerance menu selection)

**Warning**: setting the “connection tolerance” too high may result in part inaccuracies. Arc radius may also be changed if an arc endpoint is “corrected” in order to coincide with the next endpoint.

### 2.2.6 Layers

Parts are in Gsimple organised into “layers”. All bulges in a layer have the same height, use the same tools and will be machined to the same surface quality.

Layer 0 is the block. All bulges placed directly on the block (having the block as their parent) belong to layer 1. Bulges placed on these bulges (bulges that have any bulge of layer 1 as parent) belong to layer 2 and so on. The maximum number of layers is restricted only by the maximum number of bulges (32 in version 2.04).
2.2.7 Odd shaped bulge editing

Rectangular, Cylindrical and Polygonal bulges will be edited through the rectangular, cylindrical or polygonal bulge dialog. For old shaped bulges there is a special dialog with the following functions:

- Move endpoint
- Delete endpoint
- Insert endpoint
- Change edge type (line, cw arc, ccw arc)
- Change cw/ccw arc radius

Changes will not be visible until you press the OK button and the dialog is closed.

When an odd shaped bulge is being edited GSIMPLE will mark all its endpoints with a yellow empty rectangle except for endpoint 1, which will be marked with a blue empty rectangle.

If you wish to edit a rectangular, cylindrical or polygonal bulge with the odd-shapped dialog (in order for example to change a line into an arc) you will have to “explode” (Modify/Explode menu selection) the bulge. “Exploding” means declaring it to be non-special.

**NOTE:** Cylindrical bulges have two endpoints connected with two CCW arcs.

Some modification functions (for example “Turn”) will inevitably explode a “special” shaped bulge.
2.3. Pockets

A pocket is the opposite of a bulge. A pocket has always negative volume.

Pockets are colored in pink (if not selected). If they are selected “as parent” they are colored in red. If they are selected, but not “as parent”, they are colored in blue.

GSimple Version 2.04 has a restriction in the maximum number of elements (arcs and lines) a pocket may have and a restriction in the maximum number of pockets it can handle.

| Maximum number of lines and arcs or circles in a pocket | 128 |
| Maximum number of rectangular or odd shaped pockets | 64  |
| Maximum number of cylindrical pockets | 128 |
A pocket must always reside on a "parent object". Legal parent objects are the block, a bulge or another pocket. If you try to define a pocket without having already selected a parent object, GSIMPLE will respond with an error message.

In order to facilitate the user GSIMPLE includes dialogs for the two most common pocket types - rectangular, and cylindrical. Odd (arbitrary) shaped pockets can be "composed" out of any closed-line drawing.

### 2.3.1 Cylindrical pockets

The following dialog is used in order to define/edit a cylindrical pocket:

![Cylindrical Pocket Dialog](image)

Diameter and Depth expect positive numbers. Unlike cylindrical bulges, cylindrical pockets have no endpoints - they are just composed out of a circle. "Exploding" will have no effect on a cylindrical pocket.

### 2.3.2 Rectangular pockets

Rectangular pockets are similar with rectangular bulges. In rectangular bulges edge radius may be 0. In rectangular pockets it must be at least equal to the endmill radius which will be used. The rectangular pocket dialog will not allow a zero edge radius.
2.3.3 Odd shaped pockets

Odd shaped pockets are similar to odd shaped bulges. They are generally composed out of free drawings. Editing is done through the same dialog that is used for editing odd-shaped bulges.

“Exploding” a rectangular pocket will enable the user to edit it using the odd-spapped pocket dialog.

Picture: A cylindrical pocket placed into a rectangular pocket attached directly on the block
2.4 Drills

To define a drill use the following dialog, opened through the *Milling/Drill* main menu selection.

![Drill dialog](image)

Diameter and depth expect **positive** numbers. Center position X, Y is in user coordinates. A parent object (The block, a bulge or a pocket) must have already been selected in order to be able to define a drill. Chamfer and tools will be explained later on.

### 2.4.1 Composing a Drill

You can convert a free drawing circle into a drill through the *Compose/Drill* menu selection. The command will call up the same dialog with the circle data (dia and center pos) filled in. A parent object must be already selected.

![Drill composition](image)

*Picture above:* Composing a drill out of a circle with diameter 10mm placed at x=25 Y=50

*Picture below:* A drill in Isometric view. If not selected drills are colored in pink. If selected in blue.
2.5 Threads

To define a thread use the following dialog, opened through the *Milling/Thread* main menu selection.

Depth expects a **positive** number. Center position X,Y is in user coordinates. A parent object (The block, a bulge or a pocket) must have already been selected in order to be able to define a thread. Chamfer and tools will be explained later on.

### 2.5.1 Composing a Thread

You can convert a free drawing circle into a thread through the *Compose/Thread* menu selection. The command will call up the same dialog with the circle data (dia and center pos) filled in. A parent object must be already selected.

### 2.5.2 Thread type definition file

Thread types are loaded from the Thread.dat file. This is a simple (ascii) text file and may be edited through notepad or any other text editor. Fields are separated with (one or more) tabs. A typical line has 6 fields and looks like this:

```
THREADTYPE M08 8 1.25 6.8 MM
```

Field 1: Keyword (always THREADTYPE)

Field 2: Thread name (in this case M8)

Field 3: Nominal diameter in mm (in this case 8)

Field 4: Pitch in mm (in this case 1.25)

Field 5: Drill diameter in mm (in this case 6.8)

Field 6: Units mm or inch (in this case mm)
2.6 Longholes

A longhole is a simple, special pocket. In contrast to normal pockets GSIMPLE will never fine-mill longholes. A longhole cannot be used as parent - this means you cannot, for example, place a drill in a longhole.

Longholes are defined through the following dialog:

A parent object must have already been selected in order to be able to define a longhole.

Longholes are colored in pink when not selected or in blue when they are selected.

*Picture above: A longhole with diameter 10mm, length X 20mm, length Y 20mm attached on the block (the block is currently selected as parent)*
2.7 Engravings

You can use gsimple to engrave:

- Lines
- Arcs
- Circles
- Texts

Engravings are always attached on a parent object (the block, a bulge or a pocket). If you try to open an engraving dialog without having already selected the parent object (the object which will be engraved) an error message will appear asking you to select the parent object.

2.7.1 Line Engraving

To engrave a line use the following dialog:

![Line engraving dialog]

The “REL” checkbox will change the “To” to “Dist”. In this case the X and Y entries represent distances from the “FROM” position.

Both dialogs will define an “engraving line” from X10.0 Y10.0 (user coordinates) to X90.0 Y90.0

Gsimple will remember the last Engraving Width and Engraving Depth used and “propose” for the next engravings we will define.
2.7.2 Arc Engraving

To engrave an arc use the following dialog:

If the radius is too small, the OK button will be disabled (pressing it will do nothing)

2.7.3 Circle Engraving

To engrave a circle use the following dialog:

2.7.4 Text Engraving
Text engraving is done through the following dialog:

Gsimple uses a single stroke proportional (character spacing is not fixed) font for text engraving. The font is defined in a simple text (ascii) file as a sequence of lines and arcs separated (if needed) by “jumps”. Nominal character height is 12 mm. Supplied is a font file called “romansfont.cfg”. (Capital letters only - Sorry)

The character A, for example, is defined through the following lines:

```
CHARACTER A 10
JUMPTO 0 0
LINETO 4 12
LINETO 8 0
JUMPTO 2 5
LINETO 6 5
END
```

The definition starts with the CHARACTER keyword, followed by the character itself and the required space (the distance to the next character) and ends with the END keyword.

Four other keywords are related to character definition:

```
JUMPTO
LINETO
CWTO
CCWTO
```

JUMPTO and LINETO are followed by X and Y values, which represent distances from the character origin. CWTO and CCWTO are followed by X Y and RADIOUS values. The maximum number of “segments” (lines, arcs and jumps) in a character is restricted to 24.

Gsimple will resize the characters taking into account the supplied character height and the engraving width.

Spacing (%) will scale the space between the characters. Horizontal scale (%) will compress/expand the characters. Align refers to horizontal alignment and may be set to “Left”, “Right” or “Centered”.

The maximum number of texts Gsiple can handle (both engravings and simple, on screen texts) is 128.
Gsimple will engrave arcs using small lines (otherwise compression/expansion would not be possible). Two configuration variables control how many lines are going to be used.

**NCHRARCSEG** (Nominal Character Arc Segment) is the arc length, measured in degrees, corresponding to a single line. The default value is 6 degrees.

**Example**

The left parenthesis ( is defined in `romansfont.cfg` by the following lines

```
CHARACTER ( 4
JUMPTO  2  0
CWTO  2  12  10
END
```

If `NCHRARCSEG` is set to 6 (default value) and `CHRARCSEGCOR` is OFF (CHRARCSEGCOR is explained in the next paragraphs) the `CWTO` arc from 2,0 to 2,12 (approx 76 deg) will be engraved using 13 lines, since

\[ \frac{76}{6} = 12.6 \]

The number of lines is equal to the first greater or equal integer, which is the number 13. Each line will then correspond to an arc-segment of \( \frac{76}{13} = 5.84 \) degrees.

**CHRARCSEGCOR** (Character Arc Segment Correction) may be ON or OFF (default is ON).

If it is on the actual character arc segment value will be increased or decreased in accordance to the character size. To use the above example, when `CHRARCSEGCOR` is ON the left parenthesis of the previous example will be engraved with 13 lines if it is 12mm height, with 26 lines if it is 24mm height, with 52 lines if it is 48mm height etc.
The maximum number of lines that GSIMPLE may use in a character is restricted to 1024. If the engraving text is too large and CHARCSEGCOR is ON some characters may not be fully engraved. To overcome the problem increase the NCHARCSEGCOR value or turn CHARCSEGCOR OFF.

In the original gsimple files there is a second (very primitive) font definition file called stdfont.cfg. Font download is controlled by the FONT configuration variable (in gsimple.cfg file)

The line

`FONT ROMANSFONT.CFG`

will load the characters defined in the file romansfont.cfg. The line

`FONT STDFONT.CFG`

will load the characters defined in the file stdfont.cfg. The line

`FONT MYFONT.CFG`

will load the characters defined in the file myfont.cfg. The file myfont.cfg must be placed in the same folder with the executable file (gsimple.exe).

*Picture: Using stdfont.cfg. The % is not displayed because it has not been defined in stdfont.cfg*
2.8 Free drawing

Gsimple Version 2.04 has an elementary 2D CAD section. There are five basic drawing elements available:

- Lines
- Circles
- CW Arcs
- CCW Arcs
- Texts

The free drawing dialogs are similar to the engraving dialogs.

*Picture: Free drawing line dialog. The dialog is similar to the line engraving dialog.*

*Picture: Free drawing text dialog.*
2.9 Fixtures

Fixtures are simple rectangular areas on the screen. They are ignored at compilation time, so they do not affect in any way the machine code.

Fixtures are defined through the following dialog:

![Fixture dialog]

When they are not selected fixtures are colored in dark orange.

![Fixture example]

*Picture: a 16mm x 160mm x 100mm fixture at x=0, y=0*

When selected, they are colored blue.
CHAPTER 3: MILLING TOOLS AND MATERIALS

3.1 Material Groups and Subgroups

The tool manufacturers are dividing the various materials in "categories" according to the material physical properties (mainly hardness and tensile strength). These categories are called material groups and subgroups. For example St37-2 (one of the most common steels) belongs to group 1, subgroup 2.

Groups and subgroups are not fixed in gsimple. They are defined through a simple text file called amg.dat. A typical line in amg.dat looks like the following:

   1.1 Steel Magnetic soft steel, Structural steel 120 400

Fields are tab separated and they have the following meaning:

<table>
<thead>
<tr>
<th>Field</th>
<th>Meaning</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Group.Subgroup number</td>
<td>1.1</td>
</tr>
<tr>
<td>2</td>
<td>Group name</td>
<td>Steel</td>
</tr>
<tr>
<td>3</td>
<td>Subgroup name</td>
<td>Magnetic soft steel, Structural steel</td>
</tr>
<tr>
<td>4</td>
<td>Hardness (min)</td>
<td>120 (N/mm²)</td>
</tr>
<tr>
<td>5</td>
<td>Strength (max)</td>
<td>400 (N/mm²)</td>
</tr>
</tbody>
</table>

Field 4 and 5 are there for your information only, they are not used by gsimple, but must have a value, otherwise gsimple will "complain" at load time.

The "standart" material groups and subgroups used by gsimple is listed in the table in the next page. You may edit this file if you want, or even rewrite it completely in order to use any other set of material groups and subgroups you like -although it is not recommended.

3.2 Materials

There is a material list in gsimple, accessible through the Tools/Materials menu selection and the following dialog. Materials are just synonyms for a group and subgroup.
The “standart” Material Groups/Subgroups list

<table>
<thead>
<tr>
<th>Group</th>
<th>Subgroup</th>
<th>(min) N/mm²</th>
<th>(max) N/mm²</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1 Steel</td>
<td>Magnetic soft steel, Structural steel</td>
<td>120</td>
<td>400</td>
</tr>
<tr>
<td>1.2</td>
<td>Case carburizing steel</td>
<td>200</td>
<td>700</td>
</tr>
<tr>
<td>1.3</td>
<td>Plain Carbon steel</td>
<td>250</td>
<td>850</td>
</tr>
<tr>
<td>1.4</td>
<td>Alloy Steel</td>
<td>250</td>
<td>850</td>
</tr>
<tr>
<td>1.5</td>
<td>Alloy Steel, Hardened and tempered steel</td>
<td>350</td>
<td>1200</td>
</tr>
<tr>
<td>1.6</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.1 Steel</td>
<td>Free Machining Stainless steel</td>
<td>250</td>
<td>850</td>
</tr>
<tr>
<td>2.2</td>
<td></td>
<td>250</td>
<td>850</td>
</tr>
<tr>
<td>2.3</td>
<td>Ferritic+Austentic,Feritic, Martensitic</td>
<td>300</td>
<td>1000</td>
</tr>
<tr>
<td>3.1 Cast Iron</td>
<td>Lamellar graphite</td>
<td>150</td>
<td>500</td>
</tr>
<tr>
<td>3.2</td>
<td>Lamellar graphite</td>
<td>300</td>
<td>1000</td>
</tr>
<tr>
<td>3.3</td>
<td>Nodular graphite, Malleable Cast Iron</td>
<td>200</td>
<td>700</td>
</tr>
<tr>
<td>3.4</td>
<td>Nodular graphite, Malleable Cast Iron</td>
<td>300</td>
<td>1000</td>
</tr>
<tr>
<td>4.1 Titanium</td>
<td>Titanium unalloyed</td>
<td>200</td>
<td>700</td>
</tr>
<tr>
<td>4.2</td>
<td>Titanium alloyed</td>
<td>270</td>
<td>900</td>
</tr>
<tr>
<td>4.3</td>
<td>Titanium alloyed</td>
<td>350</td>
<td>1250</td>
</tr>
<tr>
<td>5.1 Nickel</td>
<td>Nickel unalloyed</td>
<td>150</td>
<td>500</td>
</tr>
<tr>
<td>5.2</td>
<td>Nickel alloyed</td>
<td>270</td>
<td>900</td>
</tr>
<tr>
<td>5.3</td>
<td>Nickel alloyed</td>
<td>350</td>
<td>1200</td>
</tr>
<tr>
<td>6.1 Copper</td>
<td>Copper</td>
<td>100</td>
<td>500</td>
</tr>
<tr>
<td>6.2</td>
<td>B-Brass, Bronze</td>
<td>200</td>
<td>700</td>
</tr>
<tr>
<td>6.3</td>
<td>w-Brass</td>
<td>200</td>
<td>700</td>
</tr>
<tr>
<td>6.4</td>
<td>High Strength Bronze</td>
<td>470</td>
<td>1500</td>
</tr>
<tr>
<td>7.1 Aluminium Magnesium</td>
<td>Al, Mg, Unalloyed</td>
<td>100</td>
<td>350</td>
</tr>
<tr>
<td>7.2</td>
<td>Al alloyed Si&lt;0.5%</td>
<td>150</td>
<td>500</td>
</tr>
<tr>
<td>7.3</td>
<td>Al alloyed 0.5%&lt;Si&lt;10%</td>
<td>120</td>
<td>400</td>
</tr>
<tr>
<td>7.4</td>
<td>Al alloyed Si&gt;10%, Whiskers reinforced, Alloyes</td>
<td>120</td>
<td>400</td>
</tr>
<tr>
<td>8.1 Synthetic Materials</td>
<td>Thermoplastics</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.2</td>
<td>Thermosetting plastics</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.3</td>
<td>Reinforced plastic materials</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.1 Hard Materials</td>
<td>Cermets (metal-ceramics)</td>
<td>550</td>
<td>1700</td>
</tr>
</tbody>
</table>
To add a new material press the NEW button, type in the material name, choose the group/subgroup it belongs to and press the OK button.

![Add new material to database](image)

**Note:** In previous versions the material list could only be changed by editing the material.dat file. From version 2.04 on, the files material.dat, tools.dat and toolmat.dat have been replaced by a single database file called gsdb.csv. This is a tab separated ascii file. You may view its contents with notepad or a spreadsheet application like excel or calc. **DO NOT MAKE ANY CHANGES OR SAVE THE FILE ON EXIT!**

### 3.3 Tool types

There are 11 different tool types in Gsimple.

- Centerdrills
- Drills
- Taps
- Thread mills
- Endmills, rough
- Endmills, fine
- Facemills, rough
- Facemills, fine
- Chamfering tools
- Engraving tools
- Special tools

For each tool Gsimple keeps two sets of data:

a. Geometry data, for example diameter, length, number of flutes or pitch in the case of taps and thread mills.
b. Milling data, for example turning speed, cutting depth and feed speed.

Geometry data are fixed. Milling data are material dependend. For this reason the tool library is able to store for every tool different milling data -one for every material group/subgroup pair.

For example Tool **DHCo10** (a 10mm DIA drill) needs to turn at 850RPM if it is used with ST37-2 (or any other material synonymus to Group/Subgroup 1.2) and at 3400RPM when used with Aluminium (Group/Subgroup 7.2)
The tool library is open—that means you are free to define your own tools, edit or delete them.

For every one of the 11 tooltypes there is a specialised set of dialogs. Below you can see, as an example, the dialogs used to define the geometry data for a tap (thread cutting tool)...

![Tool Data Base - Taps](image1)

and a rough milling face mill.

![Tool Data Base - Face Mills](image2)

If you try to use a tool for which no milling data have been defined for the block material gsimple will label it “UNSUITABLE” and ignore it at compilation time.

The maximum number of characters for a tool name is restricted to 40. Special characters (such as ! @ # % ) are not allowed in tool names. Blanks are internally converted into underscores (_). This means, underscores will be seen as blanks, the next time you open the tool library. The maximum number of tools that the library can store is 4096.
To define a new tool press the **New** button. Gsimple will open up a new dialog:

![Append/Edit Rough Face Mill Data dialog](image)

Try to use small names when defining a new tool. A good practice is to use meaningful names - for example FM21R2 for a FaceMill, 21mm dia, for Rough finishing with 2 Flutes. All data are in user units. The Dead Zone diameter is the distance between the inserts. Diameter, length and flutes are obvious. The description is restricted to 256 characters. This is a useful field for storing, for example, the tool manufacturer or the type of the inserts used.

**Example**

As an example we will add a rough milling face mill with the following data to the library:

- **Tool type:** Face Mill, Rough
- **Tool diameter:** \(d = 40\text{mm}\)
- **Tool length:** \(l = 50\text{mm}\)
- **Flutes:** \(N = 6\)
- **Dead Zone:** 30mm

![Append/Edit Rough Face Mill Data dialog](image)

Pressing the OK button will add the new facemill to the library.

The geometry data of the new tool (FM40R6) have now been defined. But the tool is still unusable because we have not provided yet any milling data.

Milling data are, as mentioned before, material dependent. The tool library may store for a given tool a set of milling parameters for every material group/subgroup pair. Of course we do not need to do this, we will just define a set of parameters for the material group/subgroups we are going to use it.

In our example we assume that we are going to use it with ST37-2 as block material. St37-2 belongs to group/subgroup 1.2, so we are going to define data just for this type of material.

www.gsimple.eu
In order to define the Milling Data we select the tool and then press the (Material Data) Mat.Data button:

We will be presented with the following dialog

In order to add milling data for a given material group we press the Add button. A list with the available material groups will be presented (see picture on the next page).

We select from the list Material Group 1.2 CASE CARBURIZING STEEL. Now there are three data fields that we have to fill. The Speed field, measured in RPM (Rounds per minute), the Feed field measured in user units (mm/min or inch/min) and the maximum cutting depth measured in user units (mm or inch).

These data are normally provided by the manufacturers - either directly through charts and tables or (most commonly) indirectly through the "Cutting Speed" and the "Feed per tooth" data.

Approximate data and calculation formulas may be found in almost all machining handbooks. In our example I will use the data provided by the Mechanical and Metal Trades Handbook, published by Verlag Europa Lehrmittel, Nourney, Vollmer GmbH & Co.KG, Dusselberger Strasse 23, 42781 Haan-Gruiter, Germany, ISBN 13 978-3-8085-1910-3
For "Low Strength Steels" the handbook gives the following data:

- Cutting Speed (Vc): $50 - 100 \text{ mm/min}$
- Feed per tooth (ft): $0.05 - 0.15 \text{ mm}$

We will use the middle values:

- $V_c = 75 \text{ mm/min}$
- $f_t = 0.10 \text{ mm}$

The tool speed is determined by the following formula:

$$n = \frac{V_c}{3.14 \times d}$$

where

- $d = 40 \text{ mm} = 0.04 \text{ m}$

Thus

$$n = \frac{75}{3.14 \times 0.04} \text{ RPM} = 597 \text{ RPM}$$

The feed rate is calculated by the following formula:

$$v_f = n \times f_t \times N$$

where

- $n = 597 \text{ RPM}$
- $f_t = 0.10 \text{ mm}$
- $N = 6$

Thus

$$v_f = 597 \times 0.10 \times 6 = 358 \text{ mm/min}$$

The cutting depth is dependent on several factors: the insert itself, the tool rigidity, the machine rigidity, the spindle power etc. Start with small values (I would suggest 1/5 to 1/3 of the insert height) and increase them as long as your machine can handle it, the vibration is low and the cut is clean.

In our case we will assume an insert 5mm high, so we will choose a cutting depth of 1.5mm

Press OK to store the data in the database (they will NOT be permanently stored until you exit gsimple)
3.4 Importing older version Tool Data Base

Goto Tools/Import Old Version Tool DataBase

The application will ask for confirmation, warning you that this action will first delete all entries from the database and then import material and tool data from the old files (material.dat, tool.dat and toolmat.dat).

Gsimple will look for these files in the program directory. If they are not found no tool will be imported.

As mentioned before, changes in the tool database will not be stored on the disk until you exit gsimple. In case of an upload of an earlier version the user will be asked, at exit, for confirmation.
CHAPTER 4: PROJECT SETUP

In order to be able to make a usable G-Code GSIMPLE must be provided with the following essential data:

- The filename of the project
- The G-Code program number and filename
- The block dimensions and material
- The part, described as a set of BULGES, POCKETS, DRILLS, THREADS and ENGRAVINGS
- The tools you are going to use

Additionally, if your machine uses a “dialect” of the standard G-Code or any other equivalent “language” you will need a set of postprocessor rules - but this topic will be discussed later on.

4.1 Defining the project filename

This is simply done with the File/Save or the File/Save As menu selections. The standard extension used for gsimple project files is .gsi

Gsimple project files are simple text files. This means you are able to open them with notepad - BUT DO NOT EDIT THEM, unless you are sure what you are doing.

A version 2.04 gsimple project file looks something like this

```
# GSIMPLE SCRIPT FILE
# www.gsimple.eu

$66 2.04
$65 0 INFO
$127 MM
$85 VF0
$12
$1 1
$2 noname.cnc
$58 1
$95 3 2.85312 543.75846 116.88111 124
$5 0 -350.000 -300.000 0.000
$96 0.000 0.000 0.000
...
...
$21 ON
$92 ON
$36 OFF 200.000 500 2 50.000
$41 OFF 3.000
$82 ON 150
$91 OFF
$125 OFF
$50 AUTO
$11 10 EM10R2 0 0.00000 0.00000
$11 11 DHCo05 0 0.00000 0.00000
$3 1 200.00000 100.00000 10.00000 0.00000 0.00000 -10.00000 0 0 -1
$4 ST37-2
$28 0.00000 0
$20 2 1 80.00000 80.00000 10.00000 110.00000 10.00000 10.00000 0.00000 0 0 -1 0 0
$24 3 1 80.00000 10.00000 50.00000 50.00000 0.00000 0 0 -1 0
$128
```

Lines starting with # are comments. All other lines (except for blanks) start with a $ sign followed by the command number, followed by the command parameters. Some should be obvious:

The line

www.gsimple.eu
$127 MM

for example means that this project was saved in a system using MM as user unit. The line

$4 ST37-2

means that the block material is ST37-2 -and so on. Some others are complicated. The line

$95 3 2.85312 543.75846 116.88111 124

for example determines the viewing parameters (TOP, ISOMETRIC SE, ISOMETRIC SW or AXONOMETRIC), the viewpoint, the zoomfactor and a number (124) coding what is visible and what not.

As a test, you can change the view-type number (the first number after $95) to 4 or 5 -these numbers correspond to front and sideview (an undocumented feature of version 2.04).

A list of all gsimple “commands” used in the configuration file, the material and tool data base, the font definition files, the thread definition files etc (gsimple uses the same syntax in all its files) can be found in Appendix I

4.2 Defining the G-Code Program Number and filename

The G-code program number and the G-code filename are defined through the Project Setup dialog.

In order to facilitate the users we have included some additional usefull informations in the Project Dialog. You may leave them blank if you wish. Fields left blank will not appear in the Printed Report (discussed later on in this manual)

You can define default values for the G-code program number and the G-code filename through the configuration file (gsimple.cfg). To edit this file (using notepad) select Setup/Configuration from the main menu.

In order to change the defaults edit the following lines in gsimple.cfg:

# Default output file (the file where the G-Code program will be stored)
GCFILE noname.cnc

# Default program number
PROGRAM 1

You will have to restart gsimple in order to activate these changes.
Project Setup Dialog

Example:

If the Output file is noname.cnc and the program number 1 gsimple will store, at compilation, the gcode in a file called noname.cnc. The first lines of the file will look something like this:

```plaintext
%01
(G-SIMPLE, VERSION 2.04)
(USER: S.KONTOGIANNIS)
(CREATED: SUN JAN 04 18:08:39 2009)
M5 G01 G28 Z0
M6 G64 G90 G98 G21
```

Besides the Output file and the Program Number the only other field that will affect the output is the Machining Center field – but this will be discussed later on, in the postprocessor section.
4.3 Defining the Block

The block (stock) is defined as described in Chapter 2.1, page 9.

4.4 Defining the end part

This end part is a set of bulges, pockets, drills, threads and engravings attached on the block or one on the other as described in detail in chapter 2.

As a reminder, please note the following

A. All bulges attached on the same object form a “layer” and must have the same height.
B. All bulges attached on objects belonging to the same layer form a layer and must have the same height.
C. Gsimple version 2.04 cannot handle islands - that is bulges attached on a pocket.

Take extreme care!

Gsimple will not check whether your part is producable or not. Placing a larger bulge on a smaller one will produce a tool crash. Placing a larger pocket in a smaller pocket will also produce a tool crash.

An example of an end part can be seen on the picture above consisting of: a Block 150x100x30mm, a rectangular Bulge 50x100x20 attached on the block, a cylindrical pocket with 80 mm diameter and 10 mm deep attached on the block and a rectangular pocket 40x80x30 mm with 10mm edge radii attached on the rectangular bulge.
4.5 Attaching tools on the toolholder

Select **Setup/Toolholder** from the main menu

You will be presented with the following dialog

To attach a tool:

I. Select a tool slot by clicking on its line
II. Press the Attach button.

Now you will be able to select any of the tools stored in the tool data base. If there is no milling data entry for the selected tool and the block material you will get a “Unsuitable” message in the remarks column.

TCS means “Through the spindle coolant”. If your machine is provided with a through the spindle cooling mechanism and there is a provision for that in the tool you can select this feature. The button functions as a toggle: pressing it with the TSC set OFF will turn it ON and vis-a-versa.
Example: attaching 8mm dia rough milling end mill in tool slot T1.

III. Select the tool type
IV. Select the desired tool
V. Press OK

4.5.1 Automatic tool assignment

In most cases the link between a milling task and a tool is obvious. If your part has one ore more M8 threads and you have attached a M8 tap in slot T3 – there is no doubt, the threadcutting will be done with tool T3.

Gsimple can generally select the proper tool for every one of the necessary milling tasks. It will silently pick up a center drill, a 6.8mm drill and a M8 tap in order to finish the M8 threads on your part. It will pick up the right end mill for a pocket and then look for an appropriate drill to predrill an insertion hole -if the end mill is not capable of plunging. It will pick up face mill to mill a cylindrical pocket (by spiralling down into the block). And if the face mills “dead-zone” is too large for this cylindrical pocket it will silently select a drill and predrill an appropriate hole in the center.

In some cases the selection is not obvious. If you attach both a M8 tap and a 1.25mm pitched thread mill on
the toolholder the selection is not any more obvious. And the same will happen if you attach both a “rough milling” end mill and a “rough milling” face mill. Which one should gsimple choose for a cylindrical pocket: the end mill or the face mill?

The answer depends on your preferences, expressed through the following configuration variables:

CONTURING
BORING
THREADING

The CONTURING variable controls the automatic selection of tools for non-cylindrical pockets. Its syntax is

CONTURING <FACEMILL> | <ENDMILL> [<ENDMILL> | <FACEMILL>]

which means that it must have at least one argument - the word FACEMILL or the word ENDMILL (values in <> must be entered as they are typed) and may have a second argument containing the same words.

The command CONTURING FACEMILL means that Gsimple will look only for facemills for rough milling bulges and (non cylindrical) pockets.

The command CONTURING ENDMILL means that Gsimple will look only for endmills.

The command CONTURING FACEMILL ENDMILL means that Gsimple will look for facemills and endmills but will prefer facemills.

The command CONTURING ENDMILL FACEMILL means the opposite - Gsimple will look for facemills and endmills but will prefer endmills.

The BORING variable controls the automatic selection of tools for cylindrical pockets. The syntax is identical to the CONTURING command:

BORING <FACEMILL> | <ENDMILL> [<ENDMILL> | <FACEMILL>]

The THREADING variable controls the automatic selection of threading tools. The syntax is

THREADING <TAP> | <THREADMILL> which means that it will accept only one argument.

In some cases even then the selection will not be obvious. For example you may have attached two rough milling endmills. In this case gsimple will try to use the one with the largest possible diameter. “Possible” means, for example, that it will not pick up a 20 mm dia endmill if your pocket contains “closed” arcs with radii smaller than 10 mm. And it will not select a 20 mm dia endmill even if there are no small arcs if it can not plunge into the block and there is no drill big enough to predrill an appropriate insertion hole.

4.5.2 Manual Tool assignment

In some cases you will want to choose the tools which are going to be used for a specific job yourself. This can be done for most of the object types, that is for

- initially milling down the block top face
- bulges
- pockets
- drills
- taps

In version 2.04 there is no manual tool selection for longholes and engravings.

The tool selection is done through the Object Dialog. In the following picture a 3 fluted, 8 mm Dia end mill is
being manually selected as the “rough milling tool” for a rectangular pocket.

![Rectangular Pocket](image)

Gsimple will let you choose an inappropriate tool for a job — but will warn you and ask you for confirmation.

![Warning](image)

But will not let you select a tool which is too short or its diameter is too big for the job.

**4.5.3 Selecting individual milling parameters**

Generally the milling parameters (speeds, feeds and cuts) are read in from the library data. In the case of automatic tool selection there is no way to change this. But gsimple will let you select individual milling parameters for every individual task if you have manually preselected a tool for this job.

To change the milling parameters:

I. Select the object by clicking on it
II. Select *Edit/Milling Params* from the main menu
Depending on the tooltype the following dialog will be opened:

Press the “Copy library data to individual” button to get a copy of the data stored in the database in the second column (which may be edited).

To select the (edited) data in the second column, labeled “individual” click the corresponding “Use” click-box and press OK. To return back to the library data select the click-box under the first column (labeled Library) and press OK.
CHAPTER 5: COMPILATION

To compile the project (make the G-Code) select G-Code/Make from the main menu.

If there are no warnings and no errors G-Simple will respond with a “Make Report” - a message like this:

```
Compilation completed successfully

Tools used:
T    Mill Time (sec)    Name
2    1158               EMCSR3

Total Milling Time: 19 min, 20 sec
```

5.1 Inspecting the G-code

You can inspect the g-code by selecting G-Code/View or G-Code/Edit from the main menu. G-Code/View will use the Internet Explorer (or the web browser you have selected through the configuration file) for viewing the code (see picture below). G-Code/Edit will use notepad or any other editor you have choosen.
5.2 Make Options

Several parameters used during compilation are controlled by the MakeOptions Dialog, accesible through the G-Code/Make Options menu selection.

All entries are in UserUnits. Most of the above fields and checkboxes are obvious. Only the non obvious will be discussed here.

"Fast block approach height" is used in drilling and taping and is the equivalent of the R-Plane in G-Code. For example, if "Fast block approach heigth" is 3 the drilling output command will become

\[ \text{G81 } \text{R3.000 } \text{Z-25.000 F85.000} \]

"Go X,Y at tool change" will be used with large object to avoid a collision between the tool and the object during toolchange. Gsimple will first move to the ordered x,y position and then change tool.

"Filter-out out-of-block paths" is a powerful filter which will eliminate from the G-Code slow running G01 paths, which are placed out of the block. This is very useful if you just want to machine just one part of an object (for example round the left edge of a long plate)
Filter-out out-of-the-block paths disabled

Filter-out out-of-the-block paths enabled
“Excess material removal” controls whether gsimple will provide a path just around the shape or will also mill away what would otherwise be left over on the block. The default setting is **ON**.

Example: A block 100x100x10 mm with a cylindrical bulge 60mm Dia x 10 mm height in the middle. Excess material removal is enabled. Below the same example with excess material removal disabled.
Disabling the Excess Material Removal may cause a tool crash. Take extreme care when disabling it!

**Drill cleaning**: If enabled GSIPLE will raise the drill to “Distance above corridor” and then turn it counterclockwise for some seconds and then stop. The procedure will be repeated as many times as you have determined.

**Finishing**: You can select between three options: fine finish vertical faces (with a special finishing tool), fine finish horizontal faces (with a special tool) or rough finishing of horizontal faces.

**Rough finishing horizontal faces**: If enabled, GSIPLE will use the same tool which it has used for rough milling to finish the horizontal faces – with speeds and feeds altered according to the parameters on the Make Options Dialog.

**Important Note**: You can select the finishing quality individually for every object through its individual dialog. For example, you can disable finishing for the project but select “fine finishing of vertical faces” for a cylindrical pocket (for fitting a ball bearing, for example).

If you disable “excess material removal” finishing of any horizontal face will be canceled - even if you have selected it.
5.3 Other options: chamfering

As you will have noticed there is a chamfer field in almost all of the objects dialog.

![Drill dialog](57x776)

You must attach a 90 deg countersink (chamfering tool) on the toolholder to use this function.

![Attached Tools dialog](57x776)

Only small numbers are allowed in the chamfer field.

In case of small drills and cylindrical bulges and if the tool diameter is big enough and the drill deep enough gsimple will just plunge the tool down. In all other cases it will contour the tool on the edges.

Gsimple will try to use the center (ir respect to its length) of the tool. If the depth is not enough it will raise the tool higher (and use a smaller diameter).
CHAPTER 6: ANIMATION

6.1 Running the compiled code

You can test the G-Code on screen through the G-Code/Run menu selection. Please note that this selection will be greyed until you have successfully compiled your project.

G-Code/Run will open up the following dialog:

```
<table>
<thead>
<tr>
<th>Tool</th>
<th>No Tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fast</td>
<td></td>
</tr>
<tr>
<td>Slow</td>
<td></td>
</tr>
</tbody>
</table>
```

Pressing the Run Button will execute one line after the other with a time delay controlled by the Fast and Slow buttons. The nominal delay is 100 milliseconds (0.1 sec). Pressing the Pause button will stop the animation. You can restart the animation by pressing again the Run button. The Single button will execute the next step.

You can start the execution from any part of the code -but you will get an error if no Program command (0nnnn) has been executed.

Fast moves are colored red. Milling moves are colored yellow. You can run the code in any view mode (top, isometric SE, isometric SW or axonometric).

The moves are not permanently drawn on the screen. Any action which forces the window to be redrawn (for example moving another window in front of it, moving it, minimizing it etc) will delete them.

www.gsimple.eu

Version 2.04
You can delete existing paths by pressing the **Clear** button.

**NOTE:** You can run the G-Code by pressing the **green** “Run G-Code” button on the right end of the upper toolbar.

If your project has not been compiled successfully you will be asked whether you want to run anyway the existing file. If it is found it will be uploaded and run.

### 6.2 Running your own code

In version 2.04 you can upload your own code and “run” it on the screen through the **G-Code/Load & Run** menu selection.

*In the picture above: animation in Isoemtric SE view.*
CHAPTER 7: THE TOOLBARS

There are two toolbars in Gsimple which will help you perform the most common tasks quickly without accessing the menu. Some functions are only available through the toolbar buttons.

The left-side toolbar is divided in four sections.

<table>
<thead>
<tr>
<th>Section</th>
<th>Function</th>
<th>Equivalent menu selection</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draw</td>
<td>Draw Line</td>
<td>Draw/Draw Line</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Circle</td>
<td>Draw/Draw Circle</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Arc</td>
<td>Draw/Draw Arc</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Rectangle</td>
<td>Draw/Draw Rectangle</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Polygon</td>
<td>Draw/Draw Polygon</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Eclipse</td>
<td>Draw/Draw Eclipse</td>
</tr>
<tr>
<td>Draw</td>
<td>Draw Text</td>
<td>Draw/Draw Text</td>
</tr>
<tr>
<td>Offset</td>
<td>(None)</td>
<td></td>
</tr>
<tr>
<td>Trim</td>
<td>(None)</td>
<td></td>
</tr>
<tr>
<td>Group</td>
<td>Draw/Group</td>
<td></td>
</tr>
<tr>
<td>Ungroup</td>
<td>Draw/Ungroup</td>
<td></td>
</tr>
<tr>
<td>Group Manager</td>
<td>(None)</td>
<td></td>
</tr>
<tr>
<td>Delete</td>
<td>Modify/Delete</td>
<td></td>
</tr>
<tr>
<td>Move</td>
<td>Modify/Move</td>
<td></td>
</tr>
<tr>
<td>Copy</td>
<td>Modify/Copy</td>
<td></td>
</tr>
<tr>
<td>Turn</td>
<td>Modify/Turn</td>
<td></td>
</tr>
<tr>
<td>Mirror</td>
<td>Modify/Mirror</td>
<td></td>
</tr>
<tr>
<td>Scale</td>
<td>Modify/Scale</td>
<td></td>
</tr>
<tr>
<td>Import DXF</td>
<td>File/Import DXF</td>
<td></td>
</tr>
<tr>
<td>Export DXF</td>
<td>File/Export DXF</td>
<td></td>
</tr>
<tr>
<td>Top View</td>
<td>View/Top View</td>
<td></td>
</tr>
<tr>
<td>Isometric SE View</td>
<td>View/Isometric SE</td>
<td></td>
</tr>
<tr>
<td>Isometric SW View</td>
<td>View/Isometric SW</td>
<td></td>
</tr>
<tr>
<td>Zoom fast in</td>
<td>View/Zoom/Zoom fast in</td>
<td></td>
</tr>
<tr>
<td>Zoom fast out</td>
<td>View/Zoom/Zoom fast out</td>
<td></td>
</tr>
<tr>
<td>Distance</td>
<td>Draw/Distance</td>
<td></td>
</tr>
<tr>
<td>Snap settings</td>
<td>Draw/Snap settings</td>
<td></td>
</tr>
</tbody>
</table>
The upper toolbar is divided in seven sections.

<table>
<thead>
<tr>
<th>Section</th>
<th>Functions</th>
<th>Equivalent menu selections</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="icons" /></td>
<td>New project&lt;br&gt;Open project&lt;br&gt;Save project&lt;br&gt;Print report</td>
<td>File/New&lt;br&gt;File/Open&lt;br&gt;File/Save&lt;br&gt;File/Print report</td>
</tr>
<tr>
<td><img src="image2.png" alt="icons" /></td>
<td>Project Setup&lt;br&gt;Block Setup&lt;br&gt;Coordinates&lt;br&gt;User Coordinates&lt;br&gt;Toolholder</td>
<td>Setup/Project&lt;br&gt;Setup/Block&lt;br&gt;Setup/Coordinates&lt;br&gt;Setup/User Coordinates&lt;br&gt;Setup/Toolholder</td>
</tr>
<tr>
<td><img src="image3.png" alt="icons" /></td>
<td>Milldown top&lt;br&gt;Rectangular bulge&lt;br&gt;Cylindrical bulge&lt;br&gt;Polygonal bulge&lt;br&gt;Compose bulge</td>
<td>Shape/Topface&lt;br&gt;Shape/Rectangular bulge&lt;br&gt;Shape/Cylindrical bulge&lt;br&gt;Shape/Polygonal bulge&lt;br&gt;Compose/Bulge</td>
</tr>
<tr>
<td><img src="image4.png" alt="icons" /></td>
<td>Rectangular pocket&lt;br&gt;Cylindrical pocket&lt;br&gt;Longhole&lt;br&gt;Compose pocket&lt;br&gt;Drill&lt;br&gt;Compose Drill&lt;br&gt;Thread&lt;br&gt;Compose thread</td>
<td>Milling/Rectangular pocket&lt;br&gt;Milling/Cylindrical pocket&lt;br&gt;Milling/Longhole&lt;br&gt;Compose/Pocket&lt;br&gt;Milling/Drill&lt;br&gt;Compose/Drill&lt;br&gt;Milling/Thread&lt;br&gt;Compose/Thread</td>
</tr>
<tr>
<td><img src="image5.png" alt="icons" /></td>
<td>Engrave line&lt;br&gt;Engrave circle&lt;br&gt;Engrave arc&lt;br&gt;Engrave text&lt;br&gt;Compose engraving</td>
<td>Engrave/Line&lt;br&gt;Engrave/Circle&lt;br&gt;Engrave/Arc&lt;br&gt;Engrave/Text&lt;br&gt;Compose/Engraving</td>
</tr>
<tr>
<td><img src="image6.png" alt="icons" /></td>
<td>Undo&lt;br&gt;Redo&lt;br&gt;Attach&lt;br&gt;Milling Parameters&lt;br&gt;Edit</td>
<td>Edit/Undo&lt;br&gt;Edit/Redo&lt;br&gt;Modify/Attach&lt;br&gt;Edit/Milling Params&lt;br&gt;Edit/Edit</td>
</tr>
<tr>
<td><img src="image7.png" alt="icons" /></td>
<td>Make G-Code&lt;br&gt;View G-Code&lt;br&gt;Transfer G-Code&lt;br&gt;Make Options&lt;br&gt;Run</td>
<td>G-Code/Make&lt;br&gt;G-Code/View&lt;br&gt;PostProc/Transfer&lt;br&gt;G-Code/Make Options&lt;br&gt;G-Code/Run</td>
</tr>
</tbody>
</table>
### APPENDIX I: GSIMPLE COMMANDS

<table>
<thead>
<tr>
<th>$n$</th>
<th>Command</th>
<th>Syntax</th>
</tr>
</thead>
</table>
| $0$ | INCLUDE | INCLUDE filename  
filename is the path to the file you want to include |
| $1$ | PROGRAM | PROGRAM [O]program-number  
program-number is a number ranging from 1 to 99999  
the 'O' is optional (for compatibility with older versions) |
| $2$ | GCFILE | GCFILE g-code-filename  
g-code-filename is the name of the file where gsimple is going to store the gcode |
| $3$ | BLOCK | BLOCK ID wx wy wz x y z stx stf quality [topface scan materialname]  
ID is the object identification number  
wx,wy,wz are the block dimensions in UU (user units) |
| $4$ | MATERIAL | MATERIAL material_name  
material_name is a name from the material list. It is a synonym for a material group/subgroup |
| $5$ | COORDINATES | COORDINATES G<54,55,56,57,58,59, 59.1,59.2,59.3> x y z [MM | INCH]  
COORDINATES <0..8> x y z [UNITS]  
COORDINATES <ANY> x y z [UNITS]  
x,y,z are the distances of the origin from the machine origin measured in MM or INCH.  
If the units are omitted x,y,z are measured in UU (user units)  
The coordinates command is used both in the configuration file (edited by the user) and the project files (written by gsimple). Gsimple uses the second syntax for the project files. |
| $6$ | LIBRARY | LIBRARY path [READONLY | CONFIRM]  
path is the path to the directory where the library files are located. The library files are:  
amg.dat - material group/subgroup definition  
threads.dat - thread definition  
gsdb.csv - tool and material database file |
| $7$ | (NOT USED) | |
| $8$ | TABLE | TABLE wx wy x y [UNITS]  
wx,wy are the dimensions of the machining center table  
x, y is the position of the bottom, left table corner measured from the machine origin.  
UNITS are either MM or INCH. If omitted UU is supposed |
| $9$ | HTSLOTS | HTSLOTS Number_of_slots Distance_between_slots Slot_width [UNITS]  
Table Horizontal (parallel to X axis) T slots |
| $10$ | VTSLOTS | VTSLOTS Number_of_slots Distance_between_slots Slot_width [UNITS]  
Table Vertical (parallel to Y axis) T slots |
<table>
<thead>
<tr>
<th>Line</th>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
</table>
| $11$ | ATTTOOL | ATTTOOL No ToolName TSC Wear_xy Wear_z  
No is the slot on the toolholder where the tool has been attached [1-99]  
Toolname is the name of the tool  
TSC is 0 or 1 depending on whether Trough The Spindle Cooland has been enabled or not  
Wear_xy and Wear_z correspond to wear values (in UU)  
Wear_z is not used |
| $12$ | PART | PART partname  
partname is a string |
| $13$ | PROJECT | PROJECT project_name |
| $14$ | DESCRIPTION | DESCRIPTION description |
| $15$ | RBULGE | RBulge ID PID WX WY WZ X Y R Chamfer TX TF Q Tolerance Scan  
Rectangular bulge |
| $16$ | BULGE | BULGE ID PID X Y WZ Chamfer TX TF Q Tolerance Scan  
Odd shaped bulge. Opens a section of LINETO, CWTO and CCWTO commands which ends with the END command |
| $17$ | END | END |
| $18$ | CBULGE | CBULGE ID PID Diameter Heigth CenterX CenterY Chamfer TX TF Q Tolerance Scan  
Cylindrical bulge |
| $19$ | POLYGON | POLYGON ID PID Edges Radius Heigth CenterX CenterY Chamfer TX TF Q Tolerance Scan  
Polygonal bulge |
| $20$ | RPOCKET | RPocket ID PID WX WY WZ X Y R Chamfer TX TF Q Tolerance Scan  
Rectangular pocket |
| $21$ | COMMENTS | COMMENTS ON | OFF | DEBUG |
| $22$ | DRILL | DRILL ID PID D L X Y Chamfer STX SPOT PD CLEAN  
Drill |
| $23$ | THREAD | THREAD ID PID D PITCH L X Y Chamfer TX TD  
Thread |
| $24$ | BORE | BORE id pid d i x y Chamfer TX TF Q Tolerance  
Cylindrical pocket |
| $25$ | (not used) | |
| $26$ | LONGHOLE | LONGHOLE ID PID D H DX DY X Y |
| $27$ | POCKET | POCKET ID PID X Y WZ Chamfer TX TF Q Tolerance Scan  
Odd shaped pocket. Opens a section of LINETO, CWTO and CCWTO commands which ends with the END command |
| $28$ | TOPFACE | Topface tf [scan]  
Mill down topface by tf  
tf in UU |
| $29   | CORRIDOR | CORRIDOR Z [<MM> | <INCH>]
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>This is the distance above the block top surface that the tool will move during rapid moves. The default value is 25 mm.</td>
</tr>
<tr>
<td>$30</td>
<td>(not used)</td>
<td></td>
</tr>
<tr>
<td>$31</td>
<td>LINETO</td>
<td>LINETO X Y</td>
</tr>
<tr>
<td>$32</td>
<td>CCWTO</td>
<td>CCWTO X Y R</td>
</tr>
<tr>
<td>$33</td>
<td>CWTO</td>
<td>CWTO X Y R</td>
</tr>
<tr>
<td>$34</td>
<td>(not used)</td>
<td></td>
</tr>
<tr>
<td>$35</td>
<td>(not used)</td>
<td></td>
</tr>
<tr>
<td>$36</td>
<td>CLEANDRILLS</td>
<td></td>
</tr>
</tbody>
</table>
| $37   | USE      | USE <G81> | <G83> | <G40> | <G41> | <G42> | <G43> | <G44> | <G12> | <G00_Before_Tool_Comp> | <G00ToEnableG43G44> | <INCH> | <MM> | <NEWFILTER> | <OLDFILTER> | <R_IN_G02_G03> | <I_J_ONLY_IN_G02_G03>
|       |          | NOTE: Some of these settings are not used any more, but are there only for backwards compatibility |
| $38   | FIXTURE  |
| $39   | DELAY    |
| $40   | OBJECTID |
| $41   | CENTERDRILLING |
| $42   | BORING   | BORING <FACEMILL> | <ENDMILL> | [ENDMILL] | [FACEMILL>]
|       |          | Controls the automatic tool assignment for cylindrical pockets. |
| $43   | CONTURING | CONTURING <FACEMILL> | <ENDMILL> | [ENDMILL] | [FACEMILL>]
|       |          | Controls the automatic tool assignment for topface milling, bulges and non cylindrical pockets. |
| $44   | WORKSPACE |
| $45   | USER     |
| $46   | HTMLVIEW |
| $47   | HELP     |
| $48   | EDIT     | EDIT path
<p>|       |          | path is the full path to your prefered text editor (used for editing the configuration file and the g-code file) |
| $49   | (NOT USED) |
| $50   | TOOLCHANGE |
| $51   | CHARACTER |
| $52   | JUMPTO   |
| $53   | ENG_TEXT |
| $54   | INFO_TEXT |
| $55   | TEXTSIZE |</p>
<table>
<thead>
<tr>
<th>Page 60</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>DWGEDITOR</strong></td>
</tr>
<tr>
<td>path is the full path to your CAD system. For example DWGEDITOR &quot;c:\Program Files\A9TECH\A9CAD\a9cad.exe&quot;</td>
</tr>
<tr>
<td><strong>DWGFILE</strong></td>
</tr>
<tr>
<td><strong>NOOFPARTS</strong></td>
</tr>
<tr>
<td><strong>ENG_CLA</strong></td>
</tr>
<tr>
<td><strong>INFO_CLA</strong></td>
</tr>
<tr>
<td><strong>QUALITY</strong></td>
</tr>
<tr>
<td><strong>PROGSTART</strong></td>
</tr>
<tr>
<td>The string will be inserted in the gcode, directly after the program-number line and the introductory comments. The ^ character will be interpreted as a new line character.</td>
</tr>
<tr>
<td><strong>PROGEND</strong></td>
</tr>
<tr>
<td>The string will be inserted in the gcode, at the end of the file. The ^ character will be interpreted as a new line character.</td>
</tr>
<tr>
<td><strong>MAGIC</strong></td>
</tr>
<tr>
<td>magic_number is the number 65918042</td>
</tr>
<tr>
<td>Used in older versions to distinguish valid project files and prevent the user from opening a file meant for another application.</td>
</tr>
<tr>
<td>Not used any more. Retained for backward compatibility.</td>
</tr>
<tr>
<td><strong>VER</strong></td>
</tr>
<tr>
<td><strong>RPLANE</strong></td>
</tr>
<tr>
<td><strong>QPLANE</strong></td>
</tr>
<tr>
<td><strong>OVERLAP</strong></td>
</tr>
<tr>
<td><strong>RFOVERLAP</strong></td>
</tr>
<tr>
<td><strong>FFOVERLAP</strong></td>
</tr>
<tr>
<td><strong>FFDZ</strong></td>
</tr>
<tr>
<td><strong>FFDXY</strong></td>
</tr>
<tr>
<td><strong>RFDZ</strong></td>
</tr>
<tr>
<td><strong>RFSOR</strong></td>
</tr>
<tr>
<td><strong>RFFOR</strong></td>
</tr>
<tr>
<td><strong>MINBTXD</strong></td>
</tr>
<tr>
<td><strong>DFTLEX</strong></td>
</tr>
<tr>
<td><strong>THREADTYPE</strong></td>
</tr>
<tr>
<td><strong>THREADING</strong></td>
</tr>
<tr>
<td><strong>CDDEPTH</strong></td>
</tr>
<tr>
<td><strong>PECKDRILL</strong></td>
</tr>
<tr>
<td><strong>CLIENT</strong></td>
</tr>
<tr>
<td><strong>FONT</strong></td>
</tr>
<tr>
<td><strong>CNC</strong></td>
</tr>
<tr>
<td><strong>OPERATOR</strong></td>
</tr>
<tr>
<td><strong>COMPANY</strong></td>
</tr>
<tr>
<td>Function</td>
</tr>
<tr>
<td>----------</td>
</tr>
<tr>
<td>TOOLHOLDER</td>
</tr>
<tr>
<td>Defines the number of tool slots for the specific machine. The maximum number is 99</td>
</tr>
<tr>
<td>LABELING</td>
</tr>
<tr>
<td>TMEFF</td>
</tr>
<tr>
<td>TRANSFER</td>
</tr>
<tr>
<td>VIEW</td>
</tr>
<tr>
<td>UCOORDS</td>
</tr>
<tr>
<td>BACKGROUND</td>
</tr>
<tr>
<td>SNAPSETTINGS</td>
</tr>
<tr>
<td>MPARAM_X</td>
</tr>
<tr>
<td>MPARAM_F</td>
</tr>
<tr>
<td>MPARAM_D</td>
</tr>
<tr>
<td>MPARAM_TC</td>
</tr>
<tr>
<td>MPARAM_TM</td>
</tr>
<tr>
<td>DEFINE</td>
</tr>
<tr>
<td>NCHRARCSEG</td>
</tr>
<tr>
<td>CHARCSEGCOR</td>
</tr>
<tr>
<td>MOVE</td>
</tr>
<tr>
<td>SELECT</td>
</tr>
<tr>
<td>UNSELECT</td>
</tr>
<tr>
<td>TURN</td>
</tr>
<tr>
<td>SCALE</td>
</tr>
<tr>
<td>MIRRORX</td>
</tr>
<tr>
<td>MIRRORY</td>
</tr>
<tr>
<td>DELETE</td>
</tr>
<tr>
<td>MSG</td>
</tr>
<tr>
<td>message is any string upto 128 characters long. The message will appear in a message box immediately on the screen when it is loaded.</td>
</tr>
<tr>
<td>UNDO</td>
</tr>
<tr>
<td>REDO</td>
</tr>
<tr>
<td>CON_TOLERANCE</td>
</tr>
<tr>
<td>REVERSE</td>
</tr>
<tr>
<td>(not used)</td>
</tr>
<tr>
<td>DEFCLAGROUP</td>
</tr>
<tr>
<td>DELCLAGROUP</td>
</tr>
<tr>
<td>ADDCLAGROUP</td>
</tr>
<tr>
<td>REMCLAGROUP</td>
</tr>
<tr>
<td>EXCESSMR</td>
</tr>
<tr>
<td>USERUNIT</td>
</tr>
<tr>
<td>$127</td>
</tr>
<tr>
<td>------</td>
</tr>
<tr>
<td>$128</td>
</tr>
<tr>
<td>$129</td>
</tr>
<tr>
<td>$130</td>
</tr>
<tr>
<td>$131</td>
</tr>
<tr>
<td>$132</td>
</tr>
<tr>
<td>$133</td>
</tr>
</tbody>
</table>