Command line arguments

Introduction

If the program is executed with `-h` or `--help` option, the following quick help will be displayed:

```
Usage:
  STLGenerator.exe [OPTION...]

  -j, --json-file arg           Project JSON file
  -o, --output-directory arg    Output directory
  -t, --material-thickness arg  Material thickness in mm (default: 0)
  -b, --box-height-3d arg       Box height in 3D in mm (default: 0)
  -s, --single-part             Merge all STL files to construct a 3D box.
                                Use --separate-top and/or --separate-bottom
                                to separate the sides
  -p, --separate-top            Do not combine the top side with the other
                                sides when using the --single-part option
  -m, --separate-bottom         Do not combine the bottom side with the
                                other sides when using the --single-part
                                option
  -y, --separate-back           Do not combine the back side with the other
                                sides when using the --single-part option
  -u, --separate-front          Do not combine the front side with the
                                other sides when using the --single-part
                                option
  -l, --separate-left           Do not combine the left side with the other
                                sides when using the --single-part option
  -g, --separate-right          Do not combine the right side with the
                                other sides when using the --single-part
                                option
  -w, --screw-size arg          Screw diameter in mm (default: 0)
  -d, --screw-head-height arg   Screw head height in mm (default: 0)
  -a, --screw-head-diameter arg
                                Screw head diameter in mm (default: 0)
  -e, --screw-extra-margin arg  Screw extra margin in mm (default: 0)
```

```
-z, --outer-sizing           Box sizes are for outside borders (inner
                             area will be smaller)
    --finger-joint           Add finger joints (box joints) to the side
                             panels so they can be assembled without
                             glue. When enabled all 6 sides default on;
                             use --no-finger-<side> to disable a specific
                             side
    --finger-size arg        Finger width in mm (0 = material thickness)
                             (default: 0)
    --finger-count arg       Number of fingers on an edge (0 = automatic
                             from finger size) (default: 0)
    --finger-clearance arg   Extra clearance added to finger slots in mm
                             (default: 0)
    --no-finger-top          Disable finger joints on the top edge
    --no-finger-bottom       Disable finger joints on the bottom edge
    --no-finger-front        Disable finger joints on the front edge
    --no-finger-back         Disable finger joints on the back edge
    --no-finger-left         Disable finger joints on the left edge
    --no-finger-right        Disable finger joints on the right edge
-r, --recess-depth arg       The depth of recess in mm (0 = no recess)
                             (default: 0)
-x, --draw-text              Whether to draw text (default: true)
-f, --font-size arg          Font size to draw in px (default: 8)
-q, --text-depth arg         Depth to engrave text in mm (default: 1)
-c, --openscad-path arg      Path to OpenSCAD executable (default:
                             C:\\Progra~1\\OpenSCAD\\openscad.exe)
-k, --keep-project-files     Do not delete OpenSCAD files at the end
-n, --only-project           Create only OpenSCAD files, do not create
                             STL files
-v, --verbose                Verbose output
-h, --help                   Print this help and exit
```

Both short and long versions of the arguments can be used.

The arguments `--json-file`, `--output-directory`, `--material-thickness`, and `--box-height-3d` are mandatory. The last two arguments can also be given in the JSON file. If both JSON and the command-line has these arguments, the command line values will be used. Other arguments have default values as mentioned above. To set an argument to false, use the long version and assign the value `false`. For example, to disable outline cut, use `--only-project=false`. Similarly, for verbose, use either `-v` or `--verbose`. Argument values can be provided with a space or an equal sign if long versions are used. For instance, both `--json-file test.json` and `--json-file=test.json`

are valid usage. For short versions, only space allowed, e.g., `-b 30`.

`--json-file`

A valid path to JSON file to be used as input. Can be a relative path (starting from the working directory) or an absolute path. You can use online visual box designer, JSON Creator or refer to MIDILLI JSON Definition document if you need help.

`--output-directory`

The directory to write the output files to. Using `.` (dot) will use the current working directory. The final files will be written into a new folder having the name of the `--json-file`.

`--material-thickness`

This value is used to calculate the depth required for creating holes for components and/or screws. The value is in millimeters (mm).

Additionally, this value is used to determine the dimensions of the left, right, front, and bottom sides (if enabled) based on whether the `--outer-sizing` parameter is active. See *Coordinate system and effects of sizing parameters* for more info.

This value is mandatory.
It can be given in the JSON (under Generic, as value "`material_thickness`") as well as in the command line. Command line will override the value if provided.

`--box-height-3d`

The height of the box in millimeters (mm). If `--outer-sizing` is used, this will be the outer height of the box. Otherwise, it is the inner height of the box.

This value is mandatory.
It can be given in the JSON (under Generic, as value "`box_height_3d`") as well as in the command line. Command line will override the value if provided.

`--single-part`

Merges all sides into a single file (`.scad` or `.stl`) to construct a complete 3D box. To separate top or bottom, use `--separate-top` and/or `--separate-bottom` respectively.

Default: false

`--separate-top`

Separate top side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Top.scad` project file in the folder.

`--separate-bottom`

Separate bottom side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Bottom.scad` project file in the folder.

`--separate-back`

Separate back side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Back.scad` project file in the folder.

`--separate-front`

Separate front side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Front.scad` project file in the folder.

`--separate-left`

Separate left side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Left.scad` project file in the folder.

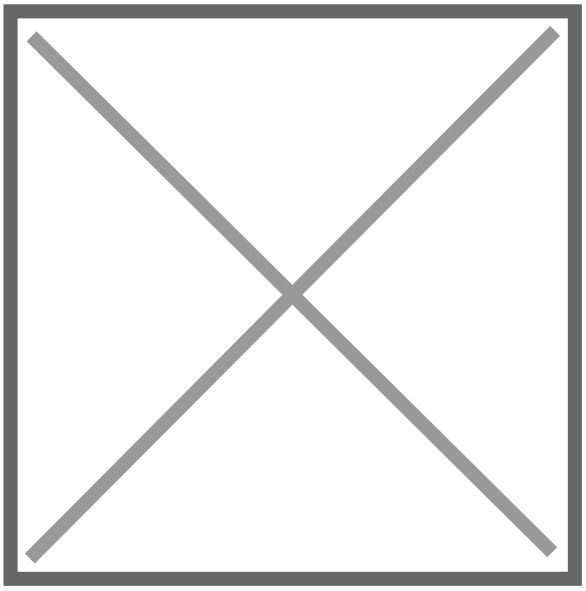
`--separate-right`

Separate right side of the box from the rest of the sides and save it as a separate file if `--single-part` option is used. This will also leave the `_Right.scad` project file in the folder.

`--screw-size`

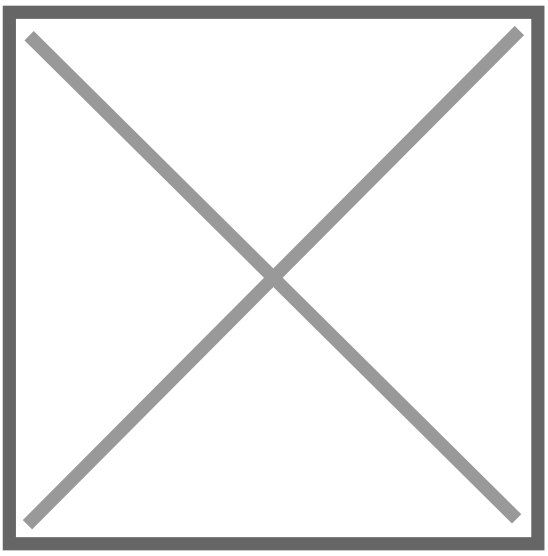
Screw hole diameter to be drawn as hole in millimeters (mm) on the top and/or bottom side of the box. Needs at least `--separate-top` or `--separate-bottom` to be used. Otherwise no screw holes will be generated.

This will also generate 4 cylindrical supports to fix the screws.



Cylindrical supports will be created if `--screw-size` is set to a value.

Default: 0 mm, no screws.

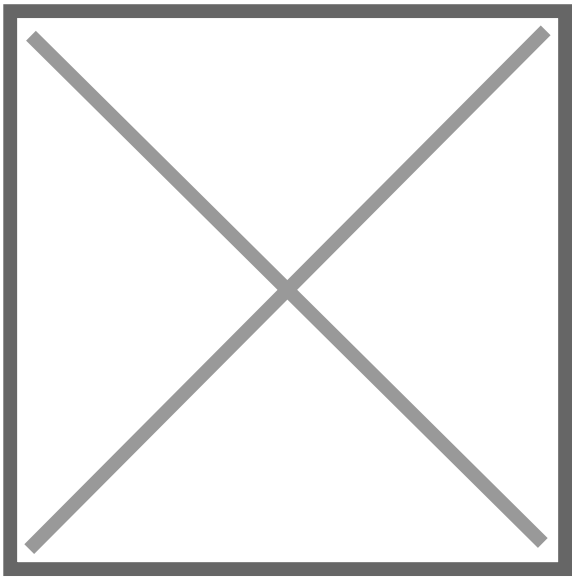


Brown: Screw diameter (or screw size)

`--screw-head-height`

The depth of the screw-head to create counterbore/recessed screws given in millimeters (mm).

Default: 0 mm, protruding screw head.

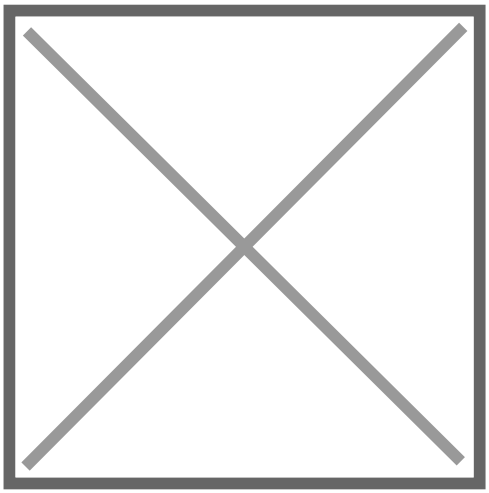


Gray: Screw head height

`--screw-head-diameter`

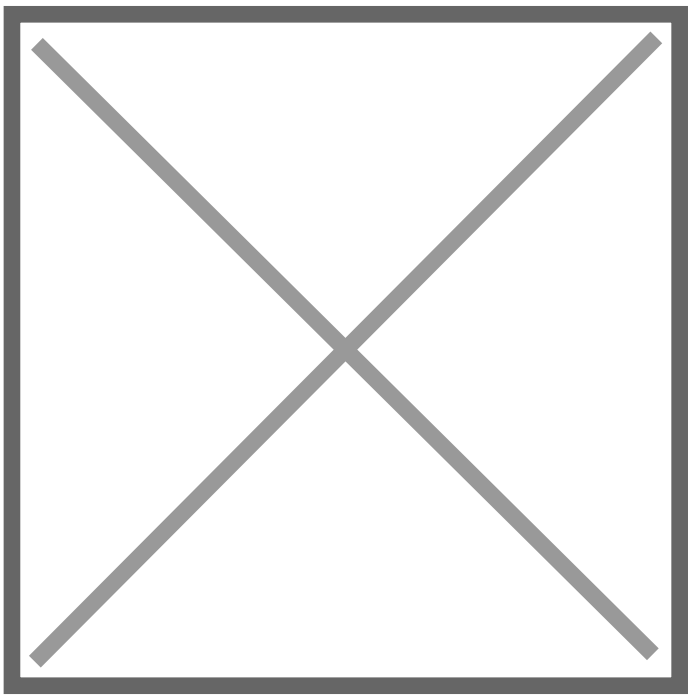
If the screws must be counterbore/recessed, any value given in millimeters (mm) (preferably greater than `--screw-size`) will create a circular shape to bury the head.

Default: 0 mm, protruding screw head.



Red: Screw head diameter

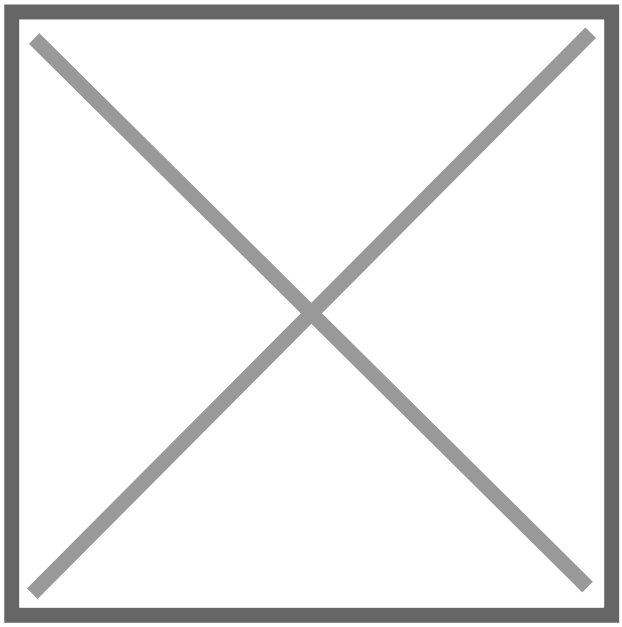
The images below show how `--screw-head-diameter` and `--screw-head-height` affect the creation of screw head related recesses.



`--screw-extra-margin`

By default, screws are added at the four corners of the top and/or bottom side of the box, leaving a margin of `--material-thickness` from the sides. If an additional margin is desired, any value given in millimeters (mm) will increase this gap.

Default: 0 mm, no additional margin.



Green: Screw extra margin

`--outer-sizing`

If used, the sizes defined in the JSON file as well as the `--material-thickness` and `--box-height-3d` arguments will be used to create the outer volume with the given dimensions. Otherwise, these values will be used to create the inner volume with the given dimensions.

The value can be given in the JSON (under Generic, as value "`outer_sizing`") as well as in the command line. Command line will override the value if provided.

Default: false, use inner dimensions.

`--recess-depth`

The depth of the recess for components in millimeters (mm).

Recesses also take the sizing from *visible_diameter* (priority) or *visible_width* and *visible_height* properties. Recesses may be helpful to have slightly lowered shapes to e.g., stabilize or limit the movement of button caps. This value does not change the behavior of the screw head recess (See `--screw-head-height`).

Default: 0 mm, no recess for the components

`--draw-text`

Whether to draw texts defined in the JSON file with `call_name`. Font size can be specified with `silkscreen_font_size` or with `--font-size` argument.

Default: true, draw text.

`--font-size`

The size of the font in pixels (px). The value is converted to OpenSCAD size value using the formula in its manual. This is only effective, if no `silkscreen_font_size` found for a `call_name`.

Default: 8 px.

`--text-depth`

The engraving depth for the text in millimeters (mm). If greater than `material-thickness`, `material-thickness` will be used. This value is used for all `call_name`s.

Default: 1 mm.

`--finger-joint`

Enables finger joints (box joints) on all 6 sides of the box so panels can be assembled without glue. By default, all 6 sides are enabled. To disable finger joints on specific sides, use the `--no-finger-<side>` arguments.

Default: false, no finger joints.

`--finger-size`

The width of each finger in millimeters (mm). If set to 0, the `--material-thickness` value will be used as the finger width.

Default: 0 mm (uses material thickness).

`--finger-count`

The number of fingers on an edge. If set to 0, the number of fingers will be calculated automatically based on the `--finger-size` value and the panel dimensions.

Default: 0 (automatic).

`--finger-clearance`

Extra clearance added to finger slots in millimeters (mm) to ensure smooth assembly.

Default: 0 mm.

`--no-finger-*`

Disable finger joints per side.

When `--finger-joint` is enabled, all 6 sides have finger joints by default. Use the following arguments to disable finger joints on specific sides:

- `--no-finger-top` — Disables finger joints on the Top panel
- `--no-finger-bottom` — Disables finger joints on the Bottom panel
- `--no-finger-front` — Disables finger joints on the Front panel
- `--no-finger-back` — Disables finger joints on the Back panel
- `--no-finger-left` — Disables finger joints on the Left panel
- `--no-finger-right` — Disables finger joints on the Right panel

`--openscad-path`

Path to OpenSCAD executable to convert `.scad` files to `.stl` files. Requires free and open-source OpenSCAD to be installed separately.

Default: C:\Program Files\OpenSCAD\openscad.exe

`--keep-project-files`

Keeps the OpenSCAD files (`.scad` files) after conversion to STL.

Default: false, remove `.scad` files after conversion to STL.

`--only-project`

Only creates `.scad` files and does not start STL conversion. This is useful e.g., to work on the files using OpenSCAD for fine-tuning or modifications.

`--verbose`

Shows more information during project creation and outputs of the OpenSCAD software during STL conversion.

`--help`

Shows the quick help with the list of arguments and their short descriptions.

Revision #8
Created 2025-11-07 10:36:23 UTC by MIDILLI
Updated 2026-08-27 09:21:53 UTC by MIDILLI