OpenSCAD Help

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Introduction

OpenSCADHelp

OpenSCAD Help Project is a compilation of help information about OpenSCAD and some libraries.

Any programming language has two parts to learn. The language syntax and himself instructions and the common libraries of constructed instructions.

OpenSCAD Help is a compilation all this help in a unique site, with a unique form of search and found help about almost the most important things of OpenSCAD.

The libraries included in this project are the next:

MCAD

Components commonly used in designing and mocking up mechanical designs https://github.com/openscad/MCAD

dotSCAD

Reduce the burden of 3D modeling in mathematics. https://github.com/JustinSDK/dotSCAD

BOSL

The Belfry OpenScad Library - A library of tools, shapes, and helpers to make OpenSCAD easier to use. https://github.com/revarbat/BOSL

What's new

2020 April 22th.

First version of PDF help file.
First version of HTML help site.
First version of EPUB help file.

Getting Started

OpenSCAD is a software for creating solid 3D CAD objects.
It is free software and available for GNU/Linux, Microsoft Windows and Mac OS X.

OpenSCAD focuses on the CAD aspects as oposition to artistic aspects. So it might be the application you are looking for when you are planning to create 3D models of machine parts.

OpenSCAD is not an interactive modeler. It is something like a 2D/3D-compiler that reads in a program file that describes the object and renders the model from this file. This gives you (the designer) full control over the modeling process. This enables you to easily change any step in the modeling process and make designs that are defined by configurable parameters.

OpenSCAD has two main operating modes:

- Preview is relatively fast using 3D graphics and the computer's GPU, but is an approximation of the model and can produce artifacts; Preview uses OpenCSG and OpenGL.
- Render generates exact geometry and a fully tessellated mesh. It is not an approximation and as such it is often a lengthy process, taking minutes or hours for larger designs. Render uses CGAL as its geometry engine.

OpenSCAD provides two types of 3D modelling:

- Constructive Solid Geometry (CSG)
- extrusion of 2D primitives into 3D space.

OpenSCAD can be downloaded from https://www.openscad.org/. **OpenSCAD** is a software for creating solid 3D CAD objects.
It is free software and available for GNU/Linux, Microsoft Windows and Mac OS X.

Unlike most free software for creating 3D models (such as the well-known application Blender), OpenSCAD does not focus on the artistic aspects of 3D modelling, but instead focuses on the CAD aspects. So it might be the application you are looking for when you are planning to create 3D models of machine parts, but probably is not what you are looking for when you are more interested in creating computer-animated movies or organic life-like models.

OpenSCAD, unlike many CAD products, is not an interactive modeler. Instead it is something like a 2D/3D-compiler that reads in a program file that describes the object and renders the model from this file. This gives you (the designer) full control over the modelling process. This enables you to easily change any step in the modelling process and make designs that are defined by configurable parameters.

OpenSCAD has two main operating modes, *Preview* and *Render*. Preview is relatively fast using 3D graphics and the computer's GPU, but is an approximation of the model and can produce artifacts; Preview uses OpenCSG and OpenGL. Render generates exact geometry and a fully tessellated mesh. It is not an approximation and as such it is often a lengthy process, taking minutes or hours for larger designs. Render uses CGAL as its geometry engine.

OpenSCAD provides two types of 3D modelling:

- Constructive Solid Geometry (CSG)
- extrusion of 2D primitives into 3D space.

Autocad DXF files are used as the data exchange format for 2D outlines. In addition to 2D paths for extrusion it is also possible to read design parameters from DXF files. Besides DXF files, OpenSCAD can read and create 3D models in the STL and OFF file formats.

OpenSCAD can be downloaded from https://www.openscad.org/. More information is available on the mailing list. OpenSCAD can also be tried online at http://openscad.net/, which is a partial port of OpenSCAD for the web.More information is available on the mailing list. OpenSCAD can also be tried online at http://openscad.net/, which is a partial port of OpenSCAD for the web.

Language Reference

The OpenSCAD Language

Introduction

Introduction

OpenSCAD is a 2D/3D and solid modeling program which is based on a Functional programming language used to create models that are previewed on the screen, and rendered into 3D mesh which allows the model to be exported in a variety of 2D/3D file formats.

A script in the OpenSCAD language is used to create 2D or 3D models. This script is a free format list of action statements.

```
object();
variable = value;
operator() action();
operator() {action(); action();}
operator() operator() {action(); action();}
operator() {operator() action();
            operator() {action(); action();}}
```

Objects

Objects are the building blocks for models, created by 2D and 3D primitives. Objects end in a semicolon ';'.

Actions

Action statements include creating objects using primitives and assigning values to variables. Action statements also end in a semicolon ';'.

Operators

Operators, or transformations, modify the location, color and other properties of objects. Operators use braces '{}' when their scope covers more than one action. More than one operator may be used for the same action or group of actions. Multiple operators are processed Right to Left, that is, the operator closest to the action is processed first. Operators do not end in semicolons ';', but the individual actions they contain do.

Examples

```
cube(5);
x = 4 + y;
rotate(40) square(5,10);
translate([10,5]) { circle(5); square(4); }
rotate(60) color("red") { circle(5); square(4); }
```

color("blue") { translate([5,3,0]) sphere(5); rotate([45,0,45]) { cylinder(10); cube([5,6,7]); } }

Comments

Comments

Comments are a way of leaving notes within the script, or code, (either to yourself or to future programmers) describing how the code works, or what it does. Comments are not evaluated by the compiler, and should not be used to describe self-evident code.

OpenSCAD uses C++-style comments:

```
// This is a comment

myvar = 10; // The rest of the line is a comment

/*
Multi-line comments
can span multiple lines.
*/
```


Values and Data Types

Values and Data Types

A value in OpenSCAD is either a Number (like 42), a Boolean (like true), a String (like "foo"), a Range (like [0: 1: 10]), a Vector (like [1,2,3]), or the Undefined value (undef).

Values can be stored in variables, passed as function arguments, and returned as function results.

[OpenSCAD is a dynamically typed language with a fixed set of data types. There are no type names, and no user defined types. Functions are not values. In fact, variables and functions occupy disjoint namespaces.]

Numbers

Numbers

Numbers are the most important type of value in OpenSCAD, and they are written in the familiar decimal notation used in other languages. Eg, -1, 42, 0.5, 2.99792458e+8.

[OpenSCAD does not support octal or hexadecimal notation for numbers.]

In additional to decimal numerals, the following names for special numbers are defined:

- **PI**

OpenSCAD has only a single kind of number, which is a 64 bit IEEE floating point number.

[OpenSCAD does not distinguish integers and floating point numbers as two different types, nor does it support complex numbers.]

Because OpenSCAD uses the IEEE floating point standard, there are a few deviations from the behavior of numbers in mathematics:

- We use binary floating point. A fractional number is not represented exactly unless the denominator is a power of 2. For example, 0.2 (2/10) does not have an exact internal representation, but 0.25 (1/4) and 0.125 (1/8) are represented exactly.
- The largest representable number is about 1e308. If a numeric result is too large, then the result can be infinity (printed as inf by echo).
- The smallest representable number is about -1e308. If a numeric result is too small, then the result can be -infinity (printed as -inf by echo).
- If a numeric result is invalid, then the result can be Not A Number (printed as **nan** by echo).
- If a non-zero numeric result is too close to zero to be representable, then the result is -0 if the result is negative, otherwise it is 0. Zero (0) and negative zero (-0) are treated as two distinct numbers by some of the math operations, and are printed differently by 'echo', although they compare as equal.

The constants **'inf'** and **'nan'** are not supported as numeric constants by OpenSCAD, even though you can compute numbers that are printed this way by 'echo'. You can define variables with these values by using:

```
inf = 1e200 * 1e200;
nan = 0 / 0;
echo(inf, nan);
```

The value 'nan' is the only OpenSCAD value that is not equal to any other value, including itself. Although you can test if a variable 'x' has the undefined value using 'x == undef', you can't use 'x == 0/0' to test if x is Not A Number. Instead, you must use 'x != x' to test if x is nan.

Boolean Values

Boolean Values

Booleans are truth values. There are two Boolean values, namely `true` and `false`. A Boolean is passed as the argument to conditional statement 'if()'. conditional operator '? :', and logical operators '!' (not), '&&' (and), and '||' (or). In all of these contexts, you can actually pass any quantity. Most values are converted to 'true' in a Boolean context, the values that count as 'false' are:

- `false`
- `0 and -0`
- `""`
- `[]`

- undef

Note that `"false"` (the string), `[0]` (a numeric vector), `[ [] ]` (a vector containing an empty vector), `[false]` (a vector containing the Boolean value false) and 0/0 (Not A Number) all count as true.

Strings

A string is a sequence of zero or more unicode characters. String values are used to specify file names when importing a file, and to display text for debugging purposes when using echo(). Strings can also be used with the text() primitive.
A string literal is written as a sequence of characters enclosed in quotation marks `"`, like this: `""` (an empty string), or `"this is a string"`.
To include a `"` character in a string literal, use `\"`. To include a `\` character in a string literal, use `\\`. The following escape sequences beginning with `\` can be used within string literals:

- `\" → "`
- `\\ → \`
- `\t → tab`
- `\n → newline`
- `\r → carriage return`
- `\u03a9 → Ω - see text() for further information on unicode characters`

Ranges

Ranges are used by for() loops and children(). They have 2 varieties:

```
[<start>:<end>]
[<start>:<increment>:<end>]
```

Although enclosed in square brackets [] , they are not vectors. They use colons : for separators rather than commas.

```
r1 = [0:10];
r2 = [0.5:2.5:20];
echo(r1); // ECHO: [0: 1: 10]
echo(r2); // ECHO: [0.5: 2.5: 20]
```

You should avoid step values that cannot be represented exactly as binary floating point numbers. Integers are okay, as are fractional values whose denominator is a power of two. For example, 0.25 (1/4) and 0.125 (1/8) are safe, but 0.2 (2/10) should be avoided. The problem with these step values is that your range may have too many or too few elements, due to inexact arithmetic.

A missing <*increment*> defaults to 1.

A range in the form [<*start*>:<*end*>] with <*start*> greater than <*end*> generates a warning and is equivalent to [<*end*>: 1: <*start*>].

A range in the form [<*start*>:1:<*end*>] with <*start*> greater than <*end*> does not generate a warning and is equivalent to [].

The <*increment*> in a range may be negative.

The Undefined Value

The Undefined Value

The undefined value is a special value written as **undef**. It's the initial value of a variable that hasn't been assigned a value, and it is often returned as a result by functions or operations that are passed illegal arguments. Finally, **`undef` can be used as a null value**, equivalent to `null` or `NULL` in other programming languages.

All arithmetic expressions containing `undef` values evaluate as `undef`.

In logical expressions, `undef` is equivalent to `false`.

Relational operator expressions with `undef` evaluate as `false` except for `undef==undef` which is `true`.

Note that numeric operations may also return 'nan' (not-a-number) to indicate an illegal argument. For example, `0/false` is `undef`, but `0/0` is 'nan'. Relational operators like < and > return `false` if passed illegal arguments. Although `undef` is a language value, 'nan' is not.

Variables

Variables

OpenSCAD variables are created by a statement with a name or **identifier**, assignment via an expression and a semicolon. The role of arrays, found in many imperative languages, is handled in OpenSCAD via vectors.

```
var = 25;
xx = 1.25 * cos(50);
y = 2 * xx + var;
logic = true;
MyString = "This is a string";
a_vector = [1,2,3];
rr = a_vector[2]; // member of vector
range1 = [-1.5:0.5:3]; // for() loop range
xx = [0:5]; // alternate for() loop range
```

OpenSCAD is a Functional programming language, as such variables are bound to expressions and keep a single value during their entire lifetime due to the requirements of referential transparency. In imperative languages, such as C, the same behavior is seen as constants, which are typically contrasted with normal variables.

In other words OpenSCAD variables are more like constants, but with an important difference. If variables are assigned a value multiple times, only the last assigned value is used in all places in the code.

See further discussion at Variables are set at compile-time, not run-time. This behavior is due to the need to supply variable input on the command line, via the use of *-D variable=value* option. OpenSCAD currently places that assignment at the end of the source code, and thus must allow a variable's value to be changed for this purpose.

Values cannot be modified during run time; all variables are effectively constants that do not change. Each variable retains its last assigned value at compile time, in line with Functional programming languages. Unlike Imperative languages, such as C, OpenSCAD is not an iterative language, and as such **the concept of *x = x + 1* is not valid.** Understanding this concept leads to understanding the beauty of OpenSCAD.

Variables can be assigned in any scope. Note that assignments are only valid within the scope in which they are defined - you are still not allowed to leak values to an outer scope. See Scope of variables for more details.

```
a=0;
if (a==0) {
   a=1; // but the value a=1 is confined to within the braces {}
}
```


Undefined variable

Undefined variable

A non assigned variable has the special value **undef**. It could be tested in conditional expression, and returned by a function.

```
Example
echo("Variable a is ", a); // Variable a is undef
if (a == undef) {
   echo("Variable a is tested undefined"); // Variable a is tested
undefined
}
```


Scope of variables

Scope of variables

When operators such as translate() and color() need to encompass more than one action (actions end in ;), braces {} are needed to group the actions, creating a new, inner scope. When there is only one semicolon, braces are usually optional.

Each pair of braces creates a new scope inside the scope where they were used. New variables can be created within this new scope. New values can be given to variables which were created

in an outer scope . These variables and their values are also available to further inner scopes created within this scope, but are **not available** to any thing outside this scope. Variables still have only the last value assigned within a scope.

```
                          // scope 1
      a = 6;              // create a
      echo(a, b);         //             6, undef
      translate([5, 0, 0]){  // scope 1.1
         a = 10;
         b = 16;          // create b
         echo(a, b);      //             100, 16 a=10; was overridden by
later a = 100;
         color("blue") {  // scope 1.1.1
            echo(a, b);   //             100, 20
            cube();
            b=20;
         }                // back to 1.1
         echo(a, b);      // 100, 16
         a = 100;         // override a in 1.1
      }                   // back to 1
      echo(a, b);         // 6, undef
      color("red"){       // scope 1.2
         cube();
         echo(a,b);       // 6, undef
      }                   // back to 1
      echo(a, b);         // 6, undef

      //In this example, scopes 1 and 1.1 are outer scopes to 1.1.1 but 1.2 is
not.
```

Anonymous scopes are not considered scopes:

```
   { angle = 45; } rotate(angle) square(10);
```

For() loops are not an exception to the rule about variables having only one value within a scope. A copy of loop contents is created for each pass. Each pass is given its own scope, allowing any variables to have unique values for that pass. No, you still can't do a=a+1;

Variables are set at compile-time, not at run-time

Variables are set at compile-time, not at run-time

Because OpenSCAD calculates its variable values at compile-time, not run-time, **the last variable assignment, within a scope apply everywhere in that scope, or inner scopes thereof**. It may be helpful to think of them as override-able constants rather than as variables.

```
      // The value of 'a' reflects only the last set value
      a = 0;
      echo(a); // 5
      a = 3;
      echo(a); // 5 a = 5;
```

While this appears to be counter-intuitive, it allows you to do some interesting things: for instance, if you set up your shared library files to have default values defined as variables at their root level, when you include that file in your own code, you can 're-define' or override those

constants by simply assigning a new value to them.

Special Variables

Special Variables

Special variables provide an alternate means of passing arguments to modules and functions. **All variables starting with a '$' are special variables**, similar to special variables in lisp. As such they are more dynamic than regular variables. (for more details see Other Language Features)

Vectors

Vectors

A vector is a sequence of zero or more OpenSCAD values. Vectors are a collection (or list or table) of numeric or boolean values, variables, vectors, strings or any combination thereof. They can also be expressions which evaluate to one of these.

Vectors handle the role of arrays found in many imperative languages. The information here also applies to lists and tables which use vectors for their data.

A vector has square brackets, [] enclosing zero or more items (elements or members), separated by commas. A vector can contain vectors, which contain vectors, etc.

examples

```
[1, 2, 3]
[a, 5, b]
[]
[5.643]
["a", "b", "string"]
[[1, r], [x, y, z, 4, 5]]
[3, 5, [6,7], [[8, 9], [10, [11, 12], 13], c, "string"]
[4/3, 6*1.5, cos(60)]
```

use in OpenSCAD:

```
cube( [width, depth, height]);      // optional spaces shown for clarity
translate( [x, y, z] )
polygon( [ [x0, y0], [x1, y1], [x2, y2] ] );
```

creation

Vectors are created by writing the list of elements, separated by commas, and enclosed in square brackets. Variables are replaced by their values.

```
cube([10, 15, 20]);
a1 = [1, 2, 3];
a2 = [4, 5];
a3 = [6, 7, 8, 9];
b = [a1, a2, a3];     // [ [1,2,3], [4,5], [6,7,8,9] ] note increased nesting depth
```

elements within vectors

Elements within vectors are numbered from 0 to n-1 where n is the length returned by len(). Address elements within vectors with the following notation:

```
e[5]          // element no 5 (sixth) at 1st nesting level
e[5][2]       // element 2 of element 5 2nd nesting level
e[5][2][0]    // element 0 of 2 of 5 3rd nesting level
e[5][2][0][1] // element 1 of 0 of 2 of 5 4th nesting level
```

example elements with lengths from len()

```
e = [ [1], [], [3, 4, 5], "string", "x", [[10, 11], [12, 13, 14], [[15, 16], [17]]] ]; // length 6
```

address	length	element
e[0]	1	[1]
e[1]	0	[]
e[5]	3	[[10,11], [12,13,14], [[15,16],[17]]]
e[5][1]	3	[12, 13, 14]
e[5][2]	2	[[15, 16], [17]]
e[5][2][0]	2	[15, 16]
e[5][2][0][1]	undef	16
e[3]	6	"string"
e[3][2]	1	"r"
s = [2,0,5];	a = 2;	
s[a]	undef	5
e[s[a]]	3	[[10, 11], [12, 13, 14], [[15, 16], [17]]]

alternate dot notation

The first three elements of a vector can be accessed with an alternate dot notation:

```
e.x     //equivalent to e[0]
e.y     //equivalent to e[1]
e.z     //equivalent to e[2]
```


Vector Operators (concat & len)

Vector Operators (concat & len)

concat

concat() combines the elements of 2 or more vectors into a single vector. No change in nesting

level is made.

```
vector1 = [1, 2, 3]; vector2 = [4]; vector3 = [5,6];
new_vector = concat(vector1, vector2, vector3); // [1, 2, 3, 4, 5, 6]

string_vector = concat("abc", "def");                 // ["abc", "def"]
one_string = str(string_vector[0], string_vector[1]); // "abcdef"
```

len

len() returns the length of vectors or strings. Indices of elements are from [0] to [length-1].

vector
- Returns the number of elements at this level.
- Single values, which are **not** vectors, return **undef**.

string
- Returns the number of characters in string.

```
a = [1, 2, 3]; echo(len(a)); // 3
```


Matrix

Matrix

A matrix is a vector of vectors.

```
Example which defines a 2D rotation matrix
mr = [
    [cos(angle), -sin(angle)],
    [sin(angle), cos(angle)]
];
```


Getting input

Getting input

Now we have variables, it would be nice to be able to get input into them instead of setting the values from code. There are a few functions to read data from DXF files, or you can set a variable with the -D switch on the command line.

Getting a point from a drawing

Getting a point is useful for reading an origin point in a 2D view in a technical drawing. The function dxf_cross reads the intersection of two lines on a layer you specify and returns the intersection point. This means that the point must be given with two lines in the DXF file, and not a point entity.

```
OriginPoint = dxf_cross(file   = "drawing.dxf",
                        layer  = "SCAD.Origin",
```

```
                                origin = [0, 0], scale = 1);
```

Getting a dimension value

You can read dimensions from a technical drawing. This can be useful to read a rotation angle, an extrusion height, or spacing between parts. In the drawing, create a dimension that does not show the dimension value, but an identifier. To read the value, you specify this identifier from your program:

```
TotalWidth = dxf_dim(file   = "drawing.dxf",
                     name   = "TotalWidth",
                     layer  = "SCAD.Origin",
                     origin = [0, 0],
                     scale  = 1);
```

For a nice example of both functions, see Example009 and the image on the homepage of OpenSCAD.

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MCAD Library

MCAD Library

This library contains components commonly used in designing and mocking up mechanical designs. It is currently unfinished and you can expect some API changes, however many things are already working.

This library was created by various authors as named in the individual files' comments. All the files are

licensed under the LGPL 2.1 (see http://creativecommons.org/licenses/LGPL/2.1/ or the included file lgpl-2.1.txt), some of them allow distribution under more permissive terms (as described in the files' comments).

Usage

You can import these files in your scripts with use <MCAD/filename.scad>, where 'filename' is one of the files listed below like 'motors' or 'servos'. Some files include useful constants which will be available with include <MCAD/filename.scad>, which should be safe to use on all included files (ie. no top level code should create geometry).

(There is a bug/feature that prevents including constants from files that "include" other files - see the openscad mailing list archives for more details. Since the maintainers aren't very responsive, may have to work around this somehow)

If you host your project in git, you can do git submodule add URL PATH in your repo to import this library as a git submodule for easy usage. Then you need to do a git submodule update --init after cloning. When you want to update the submodule, do cd PATH; git checkout master; git pull. See git help submodule for more info.

Currently Provided Tools:

regular_shapes.scad
- regular polygons, ie. 2D
- regular polyhedrons, ie. 3D

involute_gears.scad (http://www.thingiverse.com/thing:3575):
- gear()
- bevel_gear()
- bevel_gear_pair()

gears.scad (Old version):
- gear(number_of_teeth, circular_pitch OR diametrial_pitch, pressure_angle OPTIONAL, clearance OPTIONAL)

motors.scad:
- stepper_motor_mount(nema_standard, slide_distance OPTIONAL, mochup OPTIONAL)

Tools (alpha and beta quality):

nuts_and_bolts.scad: for creating metric and imperial bolt/nut holes
bearing.scad: standard/custom bearings
screw.scad: screws and augers
materials.scad: color definitions for different materials
stepper.scad: NEMA standard stepper outlines
servos.scad: servo outlines
boxes.scad: box with rounded corners
triangles.scad: simple triangles
3d_triangle.scad: more advanced triangles

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math.scad: general math functions
constants.scad: mathematical constants
curves.scad: mathematical functions defining curves
units.scad: easy metric units
utilities.scad: geometric funtions and misc. useful stuff
teardrop.scad (http://www.thingiverse.com/thing:3457): parametric teardrop module
shapes.scad: DEPRECATED simple shapes by Catarina Mota
polyholes.scad: holes that should come out well when printed

Other:

alphabet_block.scad
bitmap.scad
letter_necklace.scad
name_tag.scad
height_map.scad
trochoids.scad
libtriangles.scad
layouts.scad
transformations.scad
2Dshapes.scad
gridbeam.scad
fonts.scad
unregular_shapes.scad
metric_fastners.scad
lego_compatibility.scad
multiply.scad
hardware.scad

External utils that generate and process openscad code:

openscad_testing.py: testing code, see below
openscad_utils.py: code for scraping function names etc.

regular_shapes.scad

2D regular shapes

regular_polygon(sides, radius)

n-gons 2D shapes

triangle(radius)

pentagon(radius)

hexagon(radius)

heptagon(radius)

octagon(radius)

nonagon(radius)

decagon(radius)

hendecagon(radius)

dodecagon(radius)

ring(inside_diameter, thickness)

ellipse(width, height)

egg_outline(width, thickness)

3D regular shapes

cone

oval_prism

oval_tube

cylinder_tube

triangle_prism

triangle_tube

pentagon_prism

pentagon_tube

hexagon_prism

hexagon_tube

heptagon_prism

heptagon_tube

octagon_prism

octagon_tube

nonagon_prism

decagon_prism

hendecagon_prism

dodecagon_prism

torus

torus2

oval_torus

triangle_pyramid

square_pyramid

egg

involute_gears.scad

bevel_gear_pair()

bevel_gear()

gear()

Tests

test_gears()

test_meshing_double_helix()

test_bevel_gear()

test_bevel_gear_pair()

test_backlash()

dotSCAD Library

2D modules

arc

pie

rounded_square

line2d

polyline2d

hull_polyline2d

hexagons

polytransversals

multi_line_text

voronoi2d

3D modules

rounded_cube

rounded_cylinder

crystal_ball

line3d

polyline3d

hull_polyline3d

function_grapher

sweep

loft

starburst

voronoi3d

Transformations

along_width

hollow_out

bend

shear

2D functions

in_shape

bijection_offset

trim_shape

triangulate

contours

2D/3D functions

cross_sections

paths2sections

path_scaling_sections

bezier_surface

bezier_smooth

midpt_smooth

in_polyline

Paths

arc_path

bspline_curve

bezier_curve

helix

golden_spiral

archimedean_spiral

sphere_spiral

torus_knot

Extrusion

box_extrude

ellipse_extrude

stereographic_extrude

rounded_extrude

bend_extrude

2D Shape

shape_taiwan

shape_arc

shape_pie

shape_circle

shape_ellipse

shape_square

shape_trapezium

shape_cyclicpolygon

shape_pentagram

shape_starbusrst

shape_superformula

shape_glue2circles

shape_path_extend

2D Shape extrusions

path_extrude

ring_extrude

helix_extrude

golden_spiral_extrude

archimedean_spiral_extrude

sphere_spiral_extrude

Utils

util/sub_str

New topic

New topic

New topic

New topic

New topic

New topic

New topic

New topic

New topic

New topic

New topic

BOSL Library

Nut Job Library