yapCAD Documentation

Release unknown

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Welcome to **yapCAD**, yet another procedural CAD and computational geometry system, written in Python. This project is still in a pretty early state, though we hope you will find it useful.

Note: **yapCAD** was created to solve some fairly specific problems in procedural CAD and parametric design , and at present is most developed for generating 2D drawings in the AutoCad DXF format. If you don't know what procedural CAD or paramaterized design might be useful for, this may not be the tool for you.

On the other hand, if you are tired of manually editing your CAD files whenever you change the thickness of a material, the size of a pipe fitting, or the diameter and spacing of bolts, *etc.*, this might just be the tool you are looking for.

For an example and discussion of what parametric design is and why it might be useful see *What is Parametric Design?* below.

Much of the documentation for **yapCAD** can be found in the **README** files, as well as in the `yapcad.geom` module documentation linked below.

CHAPTER

ONE

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1.1 License

1.2 Contributors

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1.3 Changelog

1.3.1 Version 0.1.5

1.3.1.1 what’s new:

- Pre-release, heading towards V0.2.x
- Restructuring for package release
- Lots more documentation (still incomplete)
- Fixes to package configuration

1.3.1.2 Known problems

- Incomplete documentation, especially outside the `yapcad.geom` module.
- Occasional problems with complex boolean operations
- Incomplete functionality around 3D modeling
- Inconsistent inclusion of licensing boilerplate

1.3.2 Version 0.2.0

1.3.2.1 what's new:

- First announced version of **yapCAD**. Yay!
- Added new `boxcut` example, showing a fully worked (if simple) parametric design system.
- Additional documentation updates and minor bugfixes.

1.3.2.2 Known problems

- Our yapCAD readthedocs documentation is missing the expanded documentation from submodules, which is a problem since much of **yapCAD**'s documentation is in the form of docstrings in the source. I'm working on getting this sorted out. In the mean time, you may want to build a local copy of the documentation as described in the main `README` file. Or, checkout and read the source.
- Incomplete documentation, especially outside the `yapcad.geom` module.
- Occasional problems with complex boolean operations. A bug in the `intersectXY` method of the `Boolean` class.
- Incomplete functionality around 3D modeling
- Inconsistent inclusion of licensing boilerplate, other minor formatting issues.

1.4 yapcad

1.4.1 yapcad package

1.4.1.1 Submodules

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1.5 yapCAD

yet another procedural CAD and computational geometry system written in python 3

1.5.1 what's yapCAD for?

First and foremost, **yapCAD** is a framework for creating parametric, procedural, and generative design systems. You can also use **yapCAD** for other CAD, CAM, and computational geometry purposes.

1.5.2 software status

yapCAD is still very much in **beta**, although it is already being used by for professional engineering purposes. If you are using **yapCAD** in interesting ways, feel free to let us know in the yapCAD discussions forum

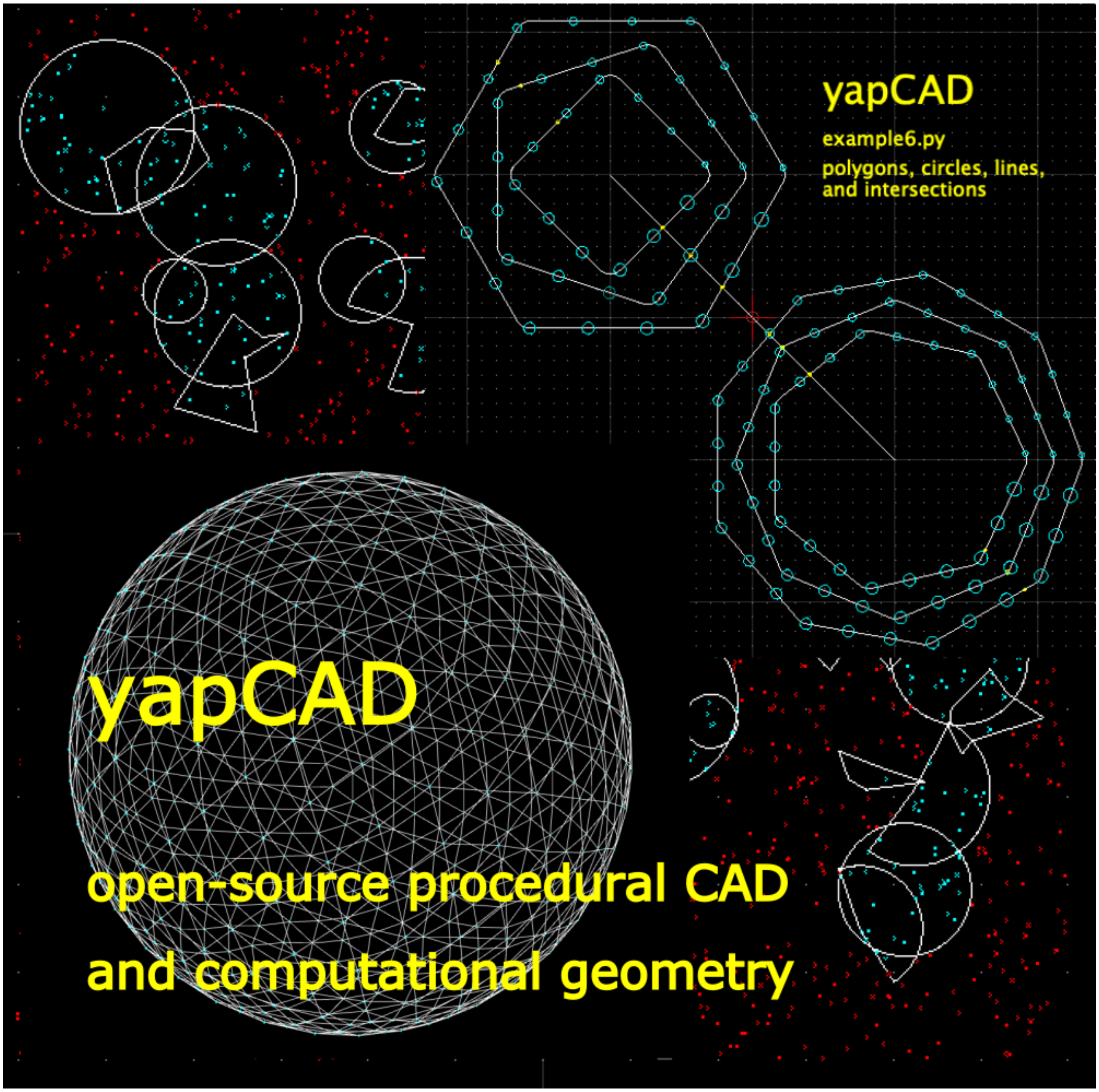

Fig. 1: **yapCAD** image

1.5.3 yapCAD installation, documentation, and examples

1.5.3.1 installation

yapCAD is a pure python library, so no special steps are required for installation. You can install it a variety of ways, but the recommended method is to use pip to install it into your local `site-packages` directory, as follows:

```
pip install yapCAD --user
```

You can also clone the github repository and install from source:

```
git clone https://github.com/rdevaul/yapCAD.git
cd yapCAD
python setup.py install --user
```

1.5.3.2 examples

The **yapCAD** github repository includes examples. To run the examples, clone the github repository as shown above, and make sure that your PYTHONPATH includes the cloned top-level `yapCAD` directory. You will find the examples in the `yapCAD/examples` directory.

For a fully worked parametric design system, see the `boxcut` example.

1.5.3.3 documentation

Online **yapCAD** documentation can be found here: https://yapcad.readthedocs.io/en/latest/ — for some reason `readthedocs.io` isn't generating the full module documentation, so you might want to build a local copy, as described below.

To build the HTML **yapCAD** documentation locally, first make sure you have the sphinx package installed:

```
pip install sphinx --user
```

Then clone the github repository as shown above, `cd` to the `yapCAD` directory, and type

```
make -C docs html
```

This will build the HTML documents in the `build/sphinx/html` directory. You can also build documentation in the other formats supported by Sphinx. See the Sphinx documentation for more information.

1.5.4 yapCAD goals

The purpose of **yapCAD** is to support 2D and 3D computational geometry and parametric, procedural, and generative design projects in python3. **yapCAD** is designed to support multiple rendering back-ends, such that a relatively small amount of code is necessary to add support for a 2D or 3D cad or drawing file format. At present, **yapCAD** supports the AutoCad DXF file format for creating two-dimensional drawings and OpenGL for creating interactive 2D and 3D renderings.

The foundations of **yapCAD** are grounded in decades of the author's experience with graphics system programming, 3D CAD and simulation. **yapCAD** has an underlying framework and architecture designed to support sophisticated computational geometry and procedural CAD applications. At the same time, the design of **yapCAD** should make easy stuff relatively easy, and the more advanced stuff possible.

The initial implementation of **yapCAD** provides DXF file creation support through the awesome ezdxf package, and interactive OpenGL visualization using the amazing pyglet package.

1.5.5 yapCAD examples

(for a more complete list, see the examples folder)

It's pretty easy to make a DXF drawing with **yapCAD**. Here is an example:

```
from yapcad.ezdxf_drawable import *
from yapcad.geom import *

#set up DXF rendering
dd=ezdxfDraw()
dd.filename = "example1-out"

## make dxf-renderable geometry

# make a point located at 10,10 in the x-y plane, rendered as a small
# red cross and circle
dd.pointstyle = 'xo' # also valid are 'x' or 'o'
dd.linecolor = 1 # set color to red (DXF index color 1)
dd.draw(point(10,10))

# make a line segment between the points -5,10 and 10,-5 in the x-y plane
# and draw it in white

dd.linecolor='white' # set color by name
dd.draw(line(point(-5,10),
             point(10,-5)))

# make an arc with a center at 0,3 with a radius of 3, from 45 degrees
# to 135 degrees, and draw it in aqua

dd.linecolor=[0,255,255] # RGB tripple, corresponds to 'aqua'
dd.draw(arc(point(0,3),3,45,135))

# write out the geometry as example1-out.dxf
dd.display()
```

The **yapCAD** system isn't just about rendering, of course, it's about computational geometry. For example, if you want to calculate the intersection of lines and arcs in a plane, we have you covered:

```
from yapcad.geom import *

# define some points
a = point(5,0)
b = point(0,5)
c = point(-3,0)
d = point(10,10)

# make a couple of lines
l1 = line(a,b)
```

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```
l2 = line(c,d)

# define a semicircular arc centered at 2.5, 2,5 with a radius of 2.5
# extending from 90 degrees to 135 degrees

arc1=arc(point(2.5,2.5),2.5,90.0,270.0)

# calculate the intersection of lines l1 and l2
int0 = intersectXY(l1,l2)

# calculate the intersection of the line l1 and the arc arc1
int1 = intersectXY(l1,arc1)

print("intersection of l1 and l2:",vstr(int0))
print("intersection of l1 and arc1:",vstr(int1))
```

And of course **yapCAD** supports calculating intersections between any simple and compound, or compound and compound geometry object.

There are lots more examples available to demonstrate the various computational geometry and rendering capabilities of **yapCAD**, including 3D geometry and OpenGL rendering.

1.5.6 yapCAD geometry

yapCAD distinguishes between "pure" geometric elements, such as lines, arcs, **etc.**, and drawn representations of those things, which might have attributes like line color, line weight, drawing layer, **etc.** This distinction is important, because the pure geometry exists independent of these attributes, which are themselves rendering-system dependent.

More importantly, for every geometric element you decide to draw, there will typically be many more — perhaps dozens — that should not be in the final rendering. By separating these two elements — computation and rendering — **yapCAD** makes them both more intentional and reduces the likelihood of certain type of drawing-quality issues, such as redundant or spurious drawing elements, that can cause confusion problems for computer-aided manufacturing (CAM).

For example, you might construct a finished drawing that includes a drill pattern that consists of circles (drill holes with centers) that follow a complex, geometrically constrained pattern. This pattern is itself the result of numerous computational geometry operations, perhaps driven by parameters relating to the size and shape of other parts.

In a program like Autodesk's Fusion360, you would typically use construction lines and constraints to create the underlying geometric pattern. These additional construction elements would have to be removed in order to make a clean DXF export of your drawing. On more than one occasion **yapCAD**'s author has created headaches by failing to remove some of these elements, confusing CAM technicians, causing delays, and sometimes resulting in expensive part fabrication errors.

Thus, **yapCAD** allows you to work freely with computational geometry without cluttering up your drawing page, since you specifically decide what to draw. It also means you can do computational geometry in **yapCAD** without ever invoking a rendering system, which can be useful when incorporating these geometry operations as part of a larger computational system, such as a tool-path generator.

As a rule, in **yapCAD** pure geometry representations capture only the minimum necessary to perform computational geometry, and the rest gets dealt with by the rendering system, which are subclasses of `Drawable` that actually make images, CAD drawings, **etc.**

1.5.6.1 vector representation in yapCAD

For the sake of uniformity, all **yapCAD** vectors are stored as projective geometry 4-vectors. (see discussion in **architecture**, below) However, most of the time you will work with them as though they are 3-vectors or 2-vectors.

It would be annoying to have to specify the redundant coordinates you aren't using every time you specify a vector, so **yapCAD** provides you with the `vect` function. It fills in defaults for the z and w parameters you may not want to specify. **e.g.**

```
>>> from yapcad.geom import *
>>> vect(10,4)
[10, 4, 0, 1]
>>> add(vect(10,4),vect(10,9))  ## add operates in 3-space
[20, 13, 0, 1.0]
```

Of course, you can specify all three (or even four) coordinates using `vect`.

Since it gets ugly to look at a bunch of [x, y, z, w] lists that all end in `0, 1]` when you are doing 2D stuff, **yapCAD** provides a convenience function `vstr` that intelligently converts **yapCAD** vectors (and lists that contain vectors, such as lines, triangles, and polygons) to strings, assuming that as long as z = 0 and w = 1, you don't need to see those coordinates.

```
>>> from yapcad.geom import *
>>> a = sub(vect(10,4),vect(10,9)) ## subtract a couple of vectors
>>> a
[0, -5, 0, 1.0]
>>> print(vstr(a)) ## pretty printing, elide the z and w coordinates
>>> [0, -5]
```

1.5.6.2 pure geometry

Pure geometric elements in **yapCAD** form the basis for computational geometry operations, including intersection and inside-outside testing. Pure geometry can also be drawn, of course — see **drawable geometry** below.

In general, **yapCAD** pure geometry supports the operations of parametric sampling, intersection calculation, inside-outside testing (for closed figures), "unsampling" (going from a point on the figure to the sampling parameter that would produce it), and bounding box calculation. **yapCAD** geometry is based on projective or homogeneous coordinates, thus supporting generalized affine transformations; See the discussion in **architecture**, below.

simple (non-compound) pure geometric elements

Simple, which is to say non-compound, geometry includes vectors, points, and lines. A vector is a list of exactly four numbers, each of which is a float or integer. A point is a vector that lies in a w > 0 hyperplane; Points are used to represent transformable coordinates in **yapCAD** geometry. A line is a list of two points.

Simple geometry also includes arcs. An arc is a list of a point and a vector, followed optionally by another point. The first list element is the center of the arc, the second is a vector in the w=-1 hyperplane (for right-handed arcs) whose first three elements are the scalar parameters `[r, s, e]`: the radius, the start angle in degrees, and the end angle in degrees. The third element (if it exists) is the normal for the plane of the arc, which is assumed to be `[0, 0, 1]` (the x-y plane) if it is not specified. Arcs are by default right-handed, but left-handed arcs are also supported, with parameter vectors lying in the w=-2 hyperplane.

compound figures

A list of more than two points represents a multi-vertex polylines. If there are at least four points in the list and the last point is the same as the first, the polyline figure is closed. (We sometimes refer to these point-list polygons or polylines as `poly()` entities.) Closed coplanar polylines are drawn as polygons and may be subject to inside-outside testing. Like other elements of pure geometry, polylines are subject to sampling, unsampling, intersection calculation, **etc.**

If instead of sharp corners you want closed or open figures with rounded corners, you should use `Polyline` or `Polygon` instances. Instances of these classes are used for representing compound geometric elements in an XY plane with C0 continuity. They differ from the point-list-based `poly()` representation in that the elements of a `Polyline` or `Polygon` can include lines and arcs as well as points. These elements need not be contiguous, as successive elements will be automatically joined by straight lines. `Polygons` are special in that they are always closed, and that any full circle elements are interpreted as "rounded corners," with the actual span of the arc calculated after tangent lines are drawn.

The `Polygon` class supports boolean operations, as described below, and also supports the `grow()` operation that makes generating a derived figure that is bigger by a fixed amount easy. This grow feature is very useful for many engineering operations, such as creating an offset path for drill holes, CAM paths, etc.

boolean operations on `Polygon` instances

yapCAD supports boolean set operations on `Polygon` instances, allowing you to construct more complex two-dimensional figures from union, intersection, and difference operations. Note that the difference operation can result in the creation of disjoint geometry in the form of two or more closed figures with positive area (see below), or closed figures with holes.

See Example 11 for a relatively simple example of boolean operations, and Example 12 for a more complex example.

yapCAD employs the convention that closed figures with right-handed geometry (increasing the sampling parameter corresponds to points that trace a counter-clockwise path) represent "positive" area, and that closed figures with left-handed geometry represent holes. This distinction is currently not operational, but will be important for future development such as turning polygons into rendered surfaces and extruding these surfaces into 3D.

disjoint compound geometry

Boolean difference operations can result in disjoint figures. It is also possible to combine **yapCAD** geometric elements in geometry lists, which is to say a list of zero or more elements of **yapCAD** pure geometry, which enforce no continuity constraints. Geometry lists provide the basis for **yapCAD** rendering.

1.5.6.3 drawable geometry

The idea is that you will do your computational geometry with "pure" geometry, and then generate rendered previews or output with one or more `Drawable` instances.

In **yapCAD**, geometry is rendered with instances of subclasses of `Drawable`, which at present include `ezdxfDrawable`, a class for producing DXF renderings using the awesome `ezdxf` package, and `pygletDrawable`, a class for interactive 2D and 3D OpenGL rendering.

To setup a drawing environment, you create an instance of the `Drawable` base class corresponding to the rendering system you want to use.

To draw, create the pure geometry and then pass that to the drawbles's `draw()` method. To display or write out the results you will invoke the `display` method of the drawable instance.

supported rendering systems

DXF rendering using `ezdxf` and interactive OpenGL rendering using `pyglet` are currently supported, and the design of **yapCAD** makes it easy to support other rendering backends.

1.5.7 yapCAD architecture

Under the hood, **yapCAD** is using projective coordinates, sometimes called homogeneous coordinates, to represent points as 3D coordinates in the w=1 hyperplane. If that sounds complicated, its because it is. :P But it does allow for a wide range of geometry operations, specifically affine transforms to be represented as composable transformation matrices. The benefits of this conceptual complexity is an architectural elegance and generality.

Support for affine transforms is at present rudimentary, but once a proper matrix transform stack is implemented it will allow for the seamless implementation and relatively easy use of a wide range of transformation and projection operations.

What does that buy you? It means that under the hood, **yapCAD** uses the same type of geometry engine that advanced CAD and GPU-based rendering systems use, and should allow for a wide range of computational geometry systems, possibly hardware-accelerated, to be built on top of it.

The good news is that you don't need to know about homogeneous coordinates, affine transforms, etc., to use **yapCAD**. And most of the time you can pretend that your vectors are just two-dimensional if everything you are doing happens to lie in the x-y plane.

So, if you want to do simple 2D drawings, we have you covered. If you want to build a GPU-accelerated constructive solid geometry system, you can do that, too.

1.5.8 Note

This project has been set up using PyScaffold 3.2.3. For details and usage information on PyScaffold see https://pyscaffold.org/.

CHAPTER TWO

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CHAPTER

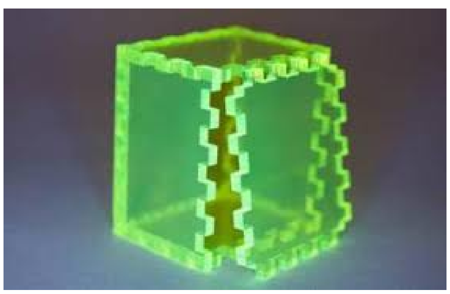
THREE

WHAT IS PARAMETRIC DESIGN?

Parametric design is a generalizable approach to solving design problems by means of parameters and algorithms, as opposed to the creation of static drawings or models. Put another way, a conventional design is like a drawing, and a parametric design is like a piece of software that you configure to create the specific drawing that you want.

3.1 the acrylic box: a parametric design example

For example, imagine that you wanted to design a decorative acrylic box to be assembled from pieces cut from a sheet of material of uniform thickness. This box will be a squeeze-fit design, so that it assembles like a 3D jigsaw puzzle, without the need for additional glue or fasteners.



You might decide on the dimensions of your box, and a scheme by which you cut the edges to create tabs and slots so that when cut, the box will fit together just so. However, the depth of your tabs and slots will necessarily depend on the thickness of the material, which might vary slightly from sheet to sheet, or vendor to vendor. Furthermore, your cutting tool (perhaps a laser cutter) will create a kerf, or width of cut, that vary from machine to machine and with depth of focus.

Finally, your box might be sized to hold a variety of contents, which themselves might vary in size and shape.

One approach is to create a conventional design. You could draw your design for one size of box, material, and thickness of cut, and then hope you have your tolerances correct. If there is a problem, or you want to change box dimensions, you will need to go back and revise your design. Each time you revise, you are essentially redoing the entire drawing from scratch.

Alternately, you could create a parametric design, in which the desired length, width, and height of the box are input parameters, along with the thickness of the material and an estimate of the kerf. Creating a parametric design system might be a bit more difficult than creating a conventional drawing, but once you are done you will be able to generate the design for any desired box, from any desired material thickness, with any kerf, simply by changing a few numbers – automatically, and without having to revise any code or drawing.

Note: For a **yapCAD** solution to this particular problem, see the `boxcut` example in the `examples` directory

This ability to solve for an entire family of related design problems with a single parametric design system is what gives this approach it's power and flexibility. For anyone who has spent hours re-drafting a drawing to accommodate minor variations in requirements, this can be an impressive force multiplier on productivity.

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