# FreeCAD Scripted Objects Modern API

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## TABLE OF CONTENTS

- FreeCAD Scripted Objects Modern API
- Preliminaries
  - Disclaimer
  - Audience
  - Goals
  - Non Goals
- Features
- General Scripted Object Architecture
  - Scripted Object Overview diagram
  - App::DocumentObject
  - Gui::ViewProvider
  - DataProxy
  - ViewProxy
  - Object creation and binding
- DataProxy Lifecycle
  - All possible states
    - Lifecycle diagram
  - State event listeners
    - on attach

- ~--<u>-</u>------
- on create
- on\_start
- on remove
- on restore
- Persistence events listeners
  - on serialize
  - on deserialize
- Active event handlers
  - on\_execute
  - on\_change
  - on\_before\_change
  - Direct property change listeners
- Other listeners
  - on\_extension
- Hooks
  - can\_link\_properties
  - is\_dirty
- Properties
  - Declaring properties
    - Property constructors
      - Examples
    - Property modes
    - Property Editor modes
    - Property change listener
    - Property access
  - Creating properties programmatically
- ViewProxy listeners
  - on\_attach
  - on\_start
  - on\_edit\_start
  - on\_edit\_end
  - on\_dbl\_click
  - on\_context\_menu
  - on\_delete
  - on\_claim\_children
  - on\_drag\_object
  - on\_drop\_object
  - on\_object\_change

- ViewProxy hooks
  - can\_drag\_objects
  - can\_drop\_objects
  - can\_drag\_object
  - can\_drop\_object
  - icon
  - Edit Modes
- Main Decorators
  - @proxy
  - @view\_proxy
  - Display Modes
- Extensions
  - Available Object Extensions
  - Available View extensions
- Migrations
  - @migrations decorator
- FreeCAD Preferences
  - Preferences API
    - Preference
    - @Preference.subscribe
- Utility functions reference
  - Global functions in fpo module
- Compatibility notes
  - Serialization and deserialization
  - Official documentation sources:
- Code examples
- Quick setup
  - Usage from a Workbench
  - Usage from Macros
  - Quick and dirty setup
  - Examples setup

## **Preliminaries**

#### **Disclaimer**

All of the following information is the result of my own research and usage of the FreeCAD's Python APIs along several years. It reflects my very own view and coding style. All the content is based on official docs, forum discussions, development of my own extensions, reading code of existing extensions and FreeCAD sources.

This document does not cover 100% of the API yet because there are still some obscure methods that can be overridden from the Python Proxies but there is no enough documentation of them, I have never used them or I have not found usage examples. My goal is to cover all of the supported features but it will take time.

#### **Audience**

This is a technical document for developers of FreeCAD extensions commonly known as Feature Python Objects or more generally Scripted Objects.

This is a python API, it is expected that the readers are python developers with a little more than basic python coding skills. The public part of the API does not use any obscure language feature but it uses classes, functions, decorators, type hints, etc...

It is also expected that the readers are FreeCAD users, and have a good understanding of the basic usage of it.

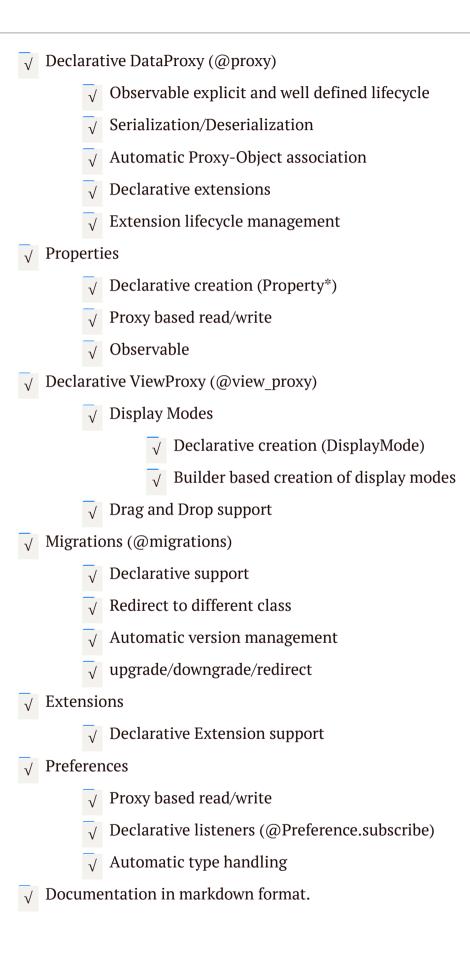
#### Goals

- The API must be developer friendly, consistent, maintainable and compatible with FC 0.20+
- The API must be an overlay on top of the existing API, so no conflicts with existing code.
- The API must be 100% documented.
- Old code and new code can be mixed, so existing projects can be upgraded gradually if desired.
- Include a tiny documentation generator to produce compact, nice and readable documentation of this API for developers.

#### **Non Goals**

- It is not intended to replace anything in the existing FreeCAD APIs.
- It is not intended to require any refactoring of existing python code.
- It is not intended to require any refactoring of existing C/C++ code.
- The documentation generator script is not for general use.

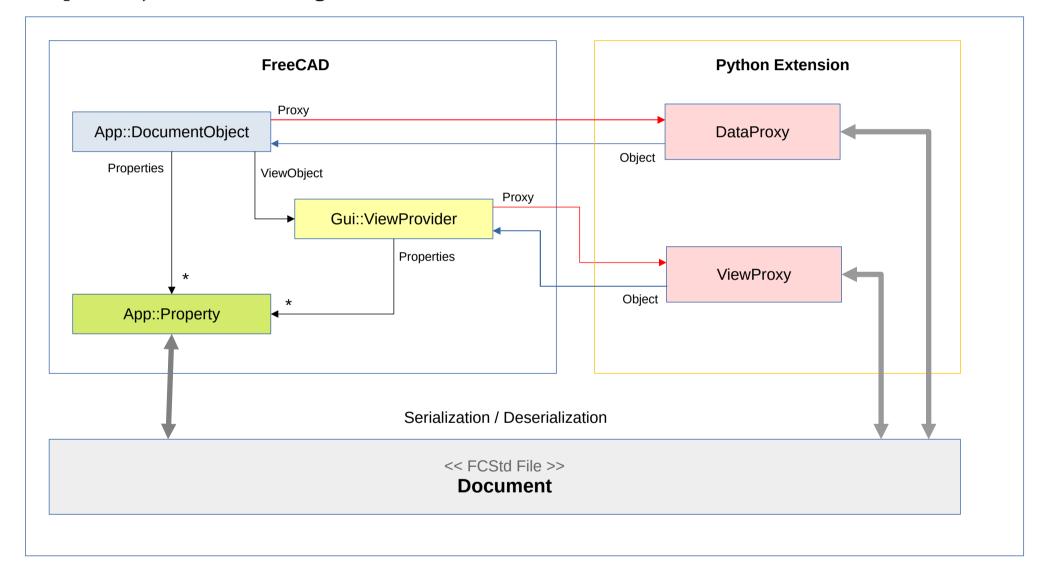
## **Features**



# General Scripted Object Architecture

Despite the widespread use of the name *FeaturePythonObject*, this is a concept and not a specific class in the Python API. Maybe a better name should be ScriptedObject. Every ScriptedObject has two main components: The Data component and the View component. Each main component is also divided in two parts: the FreeCAD object and the python Proxy object. All these 4 pieces conforms the ScriptedObject concept. It is more clear in the following diagram:

## **Scripted Object Overview diagram**



#### 

So to develop your own ScriptedObject, you need to create at least one class for the DataProxy and optionally an additional class for the ViewProxy.

#### (i) Note

View component is optional and only required for GUI part of the object.

## App::DocumentObject

Document objects are classes that define the data, geometry and logic of the features in the document. These classes are internal FreeCAD C++ classes and are instantiated from python using document.addObject(...)

See: https://wiki.freecad.org/Scripted objects

App::FeaturePython and Part::FeaturePython are the most common DocumentObject classes used to create custom objects in FreeCAD, but there are many more types supported:

- App::DocumentObjectGroupPython
- App::FeaturePython
- App::GeometryPython
- App::LinkElementPython
- App::LinkGroupPython
- App::LinkPython
- App::MaterialObjectPython
- App::PlacementPython
- Part::CustomFeaturePython
- Part::FeaturePython
- Part::Part2D0bjectPython
- Path::FeatureAreaPython
- Path::FeatureAreaViewPython
- Path::FeatureCompoundPython
- Path::FeaturePython
- Path::FeatureShapePython
- Sketcher::SketchObjectPython
- TechDraw::DrawComplexSectionPython
- TechDraw::DrawLeaderLinePython
- TechDraw::DrawPagePython
- TechDraw::DrawRichAnnoPython
- TechDraw::DrawTemplatePython
- TechDraw::DrawTilePython
- TechDraw::DrawTileWeldPython
- TechDraw::DrawViewPartPython
- TechDraw::DrawViewPython
- TechDraw::DrawViewSectionPython

- TechDraw::DrawViewSymbolPython
- TechDraw::DrawWeldSymbolPython

Source: https://wiki.freecad.org/Scripted\_objects#Available\_object\_types

#### (i) Note

There is an official class diagram, but it does not include scriptable objects apart from App::FeaturePython. See https://wiki.freecad.org/File:FreeCAD core objects.svg

#### Gui::ViewProvider

ViewProviders are classes that define the way objects will look like in the tree view and the 3D view, and how they will interact with certain graphical actions such as selection.

Source: https://wiki.freecad.org/Viewprovider

#### **DataProxy**

This is a python class responsible for managing all the data logic of your ScriptedObject, it creates the data properties and executes the required code on *document recompute*. We will see the details later.

This class is also responsible for serializing/deserializing its own internal state from/to the document.

To define a DataProxy class, just define a class and decorate it with @proxy decorator.

```
from fpo import proxy, PropertyLength, print_log

@proxy()

class MyCustomObjectProxy:
    length = PropertyLength(default=5)

def on_execute(self, obj):
    print_log("length=", self.length)
    ...

# -- usage

myCustomObjectProxy.create(name="MyThing")
```

The name of the class is irrelevant, but using *Proxy* as suffix looks like a good naming convention.

#### **ViewProxy**

This is a python class responsible for managing all the presentation logic of your ScriptedObject, it creates the presentation properties and executes the required code to display the ScriptedObject in the Tree and in the 3D scene. We will see the details later.

This class is also responsible for serializing/deserializing is own internal state from/to the document.

To define a ViewProxy class just define a class and annotate it with @view proxy decorator.

```
from fpo import view_proxy, DisplayMode

view_proxy(icon='self:my-icon.svg')

class MyCustomObjectViewProxy:
    wireframe = DisplayMode(name='Wireframe')
    shaded = DisplayMode(name='Shaded', is_default=True)

7
```

The name of the class is irrelevant, but using *ViewProxy* as suffix looks like a good naming convention.

To bind the Proxy and the ViewProxy together, you specify the ViewProxy as an argument of the @proxy decorator.

```
from fpo import proxy, view_proxy

@view_proxy()
class MyCustomViewProxy:
...

@proxy(view_proxy=MyCustomViewProxy) # <-- associate DataProxy with ViewProxy
class MyCustomObjectProxy:
...

# ----
by myCustomObjectProxy.create(name="MyThing")</pre>
```

## Object creation and binding

Once the classes are defined, you can create your objects using the create static method.

```
def create(name: str = None, label: str = None, doc: Document = None)
```

The create method takes care of adding the DocumentObject to the Document and binding the proxies, view providers, etc...

ARGUMENT	ТҮРЕ	DESCRIPTION
name	str	Internal name of the object
label	str	Label of the object used in UI.
doc	Document	Document, if omitted, current document will be used, if there is not current document, a new one will be created.

Example:

1 obj = MyCustomObjectProxy.create(name="MyThing")

# DataProxy Lifecycle

Every DataProxy object has a lifecycle. You can observe state changes using the appropriate event listeners to add your custom logic.

## All possible states

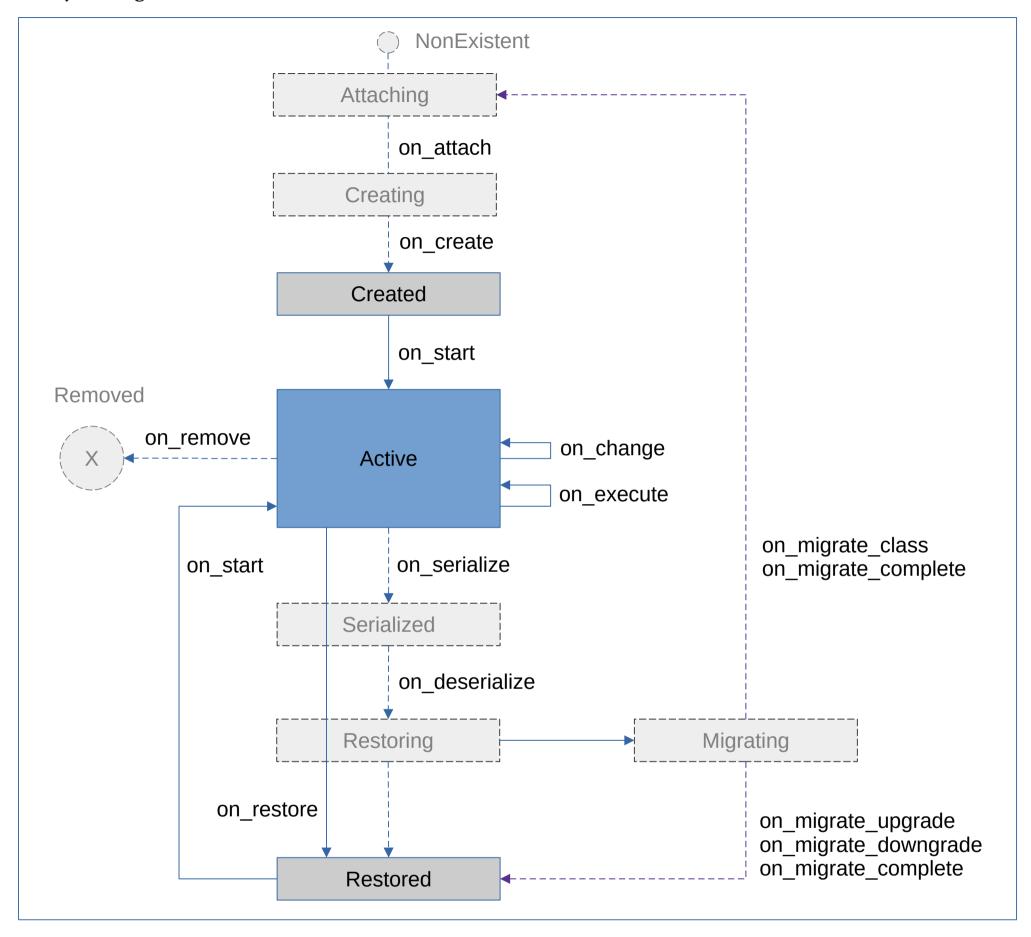
STATE	DESCRIPTION
NonExistent	(virtual) the object does not exists
Creating	(hidden) proxy instance exists but it is not initialized
Created	Proxy is created, properties and extensions are initialized
Active	Everything is initialized and the object is in consistent state. Objects and Proxies are bound together
Serialized	(virtual) the object is passivated in the FCStd file.
Restoring	(hidden) restoring everything from FCStd file.
Restored	Object is fully restored and migrations are applied if any
Removed	(virtual) object was removed from the document
Attaching	(virtual) FreeCAD is creating and binding the objects
Migrating	(virtual) Migration code is running

#### Note

Virtual states are pure conceptual, they are not present in the code but helps to understand the lifecycle. Hidden states are not observable (there are no event handlers for them)

The following diagram shows the complete lifecycle:

## Lifecycle diagram



#### **State event listeners**

To listen to a specific state change event, you create an event handler for that specific event. That is a simple method in your class with the correct signature. All listeners are of course optional.

Using state change listeners you can inject any custom logic in the right place.

Example:

```
from fpo import proxy
3 @proxy()
   class MyCustomObjectProxy:
5
6
       def on_create(self, fpo):
7
            print("My Object was created")
8
9
       def on_attach(self, fpo):
10
            print("My Object was attached to the document")
11
12
       def on_start(self, fpo):
            print("My Object is ready")
13
14
15
       def on_remove(self, fpo):
16
            print("My Object was removed")
17
18
       def on_restore(self, fpo):
            print("My Object was loaded from the file")
19
20
21
```

#### on\_attach

```
1 def on_attach(self: Proxy, fpo: DocumentObject) -> None
```

Called when the proxy is just bound to the DocumentObject and attached to the Document.

#### on create

```
1 def on_create(self: Proxy, fpo: DocumentObject) -> None
```

Called when the object is created and after all properties are created and all migrations are applied.

#### on\_start

```
1 def on_start(self: Proxy, fpo: DocumentObject) -> None
```

Called after the object is created the first time or after restored from the document. Usually any custom initialization logic must be done here.

#### on remove

```
1 def on_remove(self: Proxy, fpo: DocumentObject) -> None
```

Called before the object is removed from the Document

#### on\_restore

```
1 def on_restore(self: Proxy, fpo: DocumentObject) -> None
```

Called when the object is restored from the FCStd file

#### Persistence events listeners

FreeCAD is responsible for managing the persistence of the DocumentObjects and ViewProviders but your Proxy classes are responsible for persisting/loading its own internal state from/to the document.

## on\_serialize

```
1 def on_serialize(self: Proxy, state: Dict[str, Any]) -> None
```

This method is called to collect data from your object and store it in the document, all that you have to do is include your state into the state dictionary.

```
from fpo import proxy
3 @proxy()
   class MyCustomObjectProxy:
5
       var1: str
6
       var2: int
8
       def __init__(self, fp):
            self.var1 = 'Hello'
9
            self.var2 = 5
10
11
12
       def on_serialize(self, state: Dict[str. Any]):
            state['my_value_1'] = self.var1
13
            state['my_value_1'] = self.var2
14
15
16
```

## on\_deserialize

```
1 def on_deserialize(self: Proxy, state: Dict[str, Any]) -> None
```

This method is called to give you data from the document, all that you have to do is read the values from the state dict.

```
from fpo import proxy
2
3 @proxy()
4 class MyCustomObjectProxy:
5
       var1: str
       var2: int
6
7
       def on_deserialize(self, state: Dict[str, Any]):
8
            self.var1 = state.get('my_value_1', '')
9
            self.var2 = state.get('my_value_2', 0)
10
11
12
```

#### **Active event handlers**

When your ScriptedObject is Active, all your work is performed in on\_execute and on\_change event listeners.

#### on execute

```
1 def on_execute(self: Proxy, fpo: DocumentObject) -> None
```

This method is where you place the main scripting code of your ScriptedObject, this is called on document recompute if the object is marked as dirty (changed).

```
1 from fpo import proxy, PropertyLength
   import Part
3
4 @proxy(object_type='Part::FeaturePython')
   class CustomBoxProxy:
       width = PropertyLength(default=5.0, description='Width of the box')
6
7
       length = PropertyLength(default=5.0, description='Length of the box')
       height = PropertyLength(default=5.0, description='Height of the box')
8
9
10
       def on_execute(self, fpo):
           # Your magic happens here
11
           fpo.Shape = Part.makeBox(self.length, self.width, self.height)
12
13
14
15
16 # ----
```

```
17 obj = CustomBoxProxy.create(name='box1')
```

#### on\_change

```
def on_change(self: Proxy, fpo: DocumentObject,
    prop_name: str, new_value: Any, old_value: Any) -> None
```

Called after any property has changed.

#### on\_before\_change

```
def on_before_change(self: Proxy, fpo: DocumentObject,
prop_name: str, old_value: Any) -> None
```

Called when a property change will be performed.

## Direct property change listeners

You can listen to changes on specific properties using the @{prop}.observer decorator:

```
@proxy(object_type='Part::FeaturePython')
   class CustomBoxProxy:
       width = PropertyFloat(default=5.0, description='Width of the box')
       length = PropertyFloat(default=5.0, description='Length of the box')
4
5
       height = PropertyFloat(default=5.0, description='Height of the box')
6
7
       # Your magic happens here
8
       def on_execute(self, fpo):
9
            fpo.Shape = Part.makeBox(self.length, self.width, self.height)
10
       @length.observer
11
       def length_changed(self, new_value, old_value):
12
            print(f"Hey! length has changed from {old_Value} to {new_value}")
13
14
```

#### Other listeners

#### on\_extension

```
1 def on_extension(self: Proxy, fpo: DocumentObject, extension: str) -> None
```

Called when an extension is added to the object. Extensions add predefined behaviors to the DocumentObject, making it behave like a group, link, attachable, etc... see: extensions

## Hooks

More optional methods called by FreeCAD to get some info from the Proxy

## can\_link\_properties

```
1 def can_link_properties(self) -> bool
```

Return True if your Proxy accepts linked properties

**i** Note

to be confirmed as official documentation is not clear about this.

## is\_dirty

```
1 def is_dirty(self) -> bool
```

Return True if your Proxy in a state that requires recompute

# **Properties**

The main interaction between your ScriptedObject and the user is by managing property values, the user sets the property values and you do something useful with that.

Properties are declared using special property constructors, there is one constructor per property type.

For a reference of all property types, check the official docs:

• https://wiki.freecad.org/FeaturePython Custom Properties

## **Declaring properties**

Each property can be declared with a proxy attribute.

For example, to create an Integer property:

```
1 @proxy()
2 class MyMagicProxy:
3    my_property = PropertyInteger(section="Basic", default=5)
4    # Optional listener
6    @my_property.observer
7    def my_property_obs(self, new_value, old_value):
8         print(f"my_property has changed from {old_value} to {new_value}")
9
```

#### **Property constructors**

```
def Property{__property_type__}}(
2
           name: str = None,
            section: str = 'Data',
3
4
           default: Any = None,
           description: str = '',
            mode: PropertyMode = PropertyMode.Default,
6
7
            observer_func: Callable = None,
            link_property: str = None,
8
9
            enum: Enum = None,
            options: Callable[[], List[str]] = None)
10
```

ARGUMENT	DESCRIPTION
name	Name of the property, deduced from the attribute if missing
section	Sub section in the property editor
default	Default value of the property
description	Tooltip text
mode	A combination of PropertyMode flags
enum	Only valid for PropertyEnumeration. The enum type.
options	Only valid for PropertyOptions . A function that returns the list of options
link_property	Key of the Link property (see extensions) App::LinkExtensionPython
observer_func	Function to listen for property changes. You can also use the observer decorator.

## Examples

```
from fpo import PropertyInteger, PropertyLength, PropertyAngle, proxy

@proxy()

class MyProxy:
    x = PropertyInteger(default=5)
    y = PropertyLength(default=0)
    w = PropertyAngle(default=30)
```

## Property modes

On property declaration, you can specify a mode or a combination of them. Supported modes are the following:

MODE	DESCRIPTION
PropertyMode.Default	No special property type
PropertyMode.ReadOnly	Property is read-only in the editor
PropertyMode.Transient	Property won't be saved to file
PropertyMode.Hidden	Property won't appear in the editor
PropertyMode.Output	Modified property doesn't touch its parent container
PropertyMode.NoRecompute	Modified property doesn't touch its container for recompute
PropertyMode.NoPersist	Property won't be saved to file at all

## **Property Editor modes**

Editor modes for properties are different than actual property modes and are transient:

MODE	DESCRIPTION
PropertyEditorMode.Default	read/write access in the editor
PropertyEditorMode.ReadOnly	Property is read-only in the editor
PropertyEditorMode.Hidden	Property won't appear in the editor

## Property change listener

Any function can be subscribed to the property to listen for change events. The arguments of the property listener are all optional

```
1 @proxy()
   class MyMagicProxy
3
       my_prop1 = PropertyInteger(section="Basic", default=5)
4
       my_prop2 = PropertyInteger(section="Basic", default=5)
5
       my_prop3 = PropertyInteger(section="Basic", default=5)
6
       @my_prop1.observer
7
8
       def listener1(self, new_value, old_value):
            print(f"my_property1 has changed from {old_value} to {new_value}")
9
10
11
       @my_prop2.observer
12
       def listener2(self, new_value):
13
            print(f"my_property2 has changed {new_value}")
14
       @my_prop3.observer
15
       def listener3(self):
16
            print(f"my_property3 has changed")
17
18
```

#### Important

Only one observer (listener) method can be attached to each property.

#### Property access

All properties can be accessed from the Proxy object using the declared property name. It is internally proxyfied to the actual DocumentObject.

```
@proxy()
   class MyMagicProxy
       my_property1 = PropertyInteger(section="Basic", default=5)
4
       def on_execute(self, fp):
5
6
           # Transparently access the property from the remote object
 7
8
           x = self.my_property1
9
           # Transparently update the property from the remote object
10
11
            self.my_property1 = 10
12
```

#### Note

Properties are only proxies of the actual properties in the internal FreeCAD object (DocumentObject). So persistence is managed by FreeCAD. additional state of your proxy object must be serialized/deserialized by you in on\_serialize / on\_deserialize listeners.

#### **Creating properties programmatically**

It is also possible to create properties programmatically using the direct API, but in that case you manage them directly from the object.

```
@proxy()
   class MyMagicProxy
3
       def on_start(self, fp):
4
            if not hasattr(fp, 'Length'):
5
                fp.addProperty(
6
                    "App::PropertyLength",
                    "Length", "Box", "Length of the box").Length = 1.0
8
9
       def on_execute(self, fp):
10
            # read
11
12
            x = fp.Length
13
            # write
            fp.Length = 10
14
15
```

# ViewProxy listeners

ViewProxy has a lightly lifecycle compared to DataProxy but has a lot of listeners and methods to interface with FreeCAD GUI.

#### on\_attach

```
1 def on_attach(self, vp: ViewObject) -> None
```

Called when the ViewObject is attached to the Document. Usually init logic is here.

#### on\_start

```
1 def on_start(self, vp: ViewObject) -> None
```

Called when the ViewObject is attached to the Document and all declared properties are created and all declared Display Modes are created.

## on\_edit\_start

```
1 def on_edit_start(self, vp: ViewObject, mode: EditMode = EditMode.Default) -> None
```

Called when the user request edit. See edit modes

#### on\_edit\_end

```
1 def on_edit_end(self, vp: ViewObject, mode: EditMode = EditMode.Default) -> None
```

Called when the user terminates editing. See edit modes

## on\_dbl\_click

```
1 def on_dbl_click(self, vp: ViewObject) -> None
```

Called when the user double clicks the Tree Node.

#### on\_context\_menu

```
1 def on_context_menu(self, vp: ViewObject, menu: QMenu) -> None
```

Called to populate the context menu. You can add actions to the menu object:

```
def on_context_menu(self, vp, menu):
menu.addAction(...)

4
```

## on\_delete

```
1 def on_delete(self, vp: ViewObject, sub_elements) -> None
```

Called when the ViewObject is deleted. Usually to re-expose the child nodes.

## on\_claim\_children

```
1 def on_claim_children(self) -> List[DocumentObject]
```

Returns a list of Document Objects that need to be shown as child nodes of this ScriptedObject.

## on\_drag\_object

```
1 def on_drag_object(self, vp: ViewObject, obj: DocumentObject) -> None
```

Called if the obj was allowed to be dragged. You perform the drag logic here.

## on\_drop\_object

```
1 def on_drop_object(self, vp: ViewObject, obj: DocumentObject) -> None
```

Called if the dropped obj was accepted. You perform the drop logic here.

## on\_object\_change

```
1 def on_object_change(self, fp: DocumentObject, prop_name: str) -> None
```

Called when a property changes on the associated DocumentObject (Not the ViewObject).

# ViewProxy hooks

## can\_drag\_objects

```
1 def can_drag_objects(self) -> bool
```

Returns True if this VP accepts dragging of sub-elements

## can\_drop\_objects

```
1 def can_drop_objects(self) -> bool
```

Returns True if this VP accepts dropping of sub-elements

## can\_drag\_object

```
1 def can_drag_object(self, obj: DocumentObject) -> bool
```

Returns True if this VP accepts dragging of the dragged obj

## can\_drop\_object

```
1 def can_drop_object(self, obj: DocumentObject) -> bool
```

Returns True if this VP accepts dropping of the incoming obj

#### icon

```
1 def icon(self) -> str
```

Returns the path of the icon (Tree Node Icon). If the returned value is prefixed with 'self:' the path will be resolved relatively to the file where the class is declared.

#### **Edit Modes**

MODE	DESCRIPTION
Default(0)	The object will be edited using the mode defined internally to be the most appropriate for the object type
Transform(1)	The object will have its placement editable with the Std TransformManip command
Cutting(2)	This edit mode is implemented as available but currently does not seem to be used by any object
Color(3)	The object will have the color of its individual faces editable with the Part FaceColors command

# **Main Decorators**

There are two entry points for the API, the DataProxy and the ViewProxy. Both of them are created decorating a class with the corresponding decorators @proxy and @view\_proxy respectively.

## @proxy

```
1 @proxy(
2    object_type: str = 'App::FeaturePython',
3    subtype: str = None,
4    view_proxy: ViewProxy = None,
5    extensions: Iterable[str] = None,
6    view_provider_name_override: str = None,
7    version: int = 1)
```

Converts a user defined class into a full blown DataProxy with all of the lifecycle management, versioning, proxyfied properties, extensions, etc...

ARGUMENT	DESCRIPTION
object_type *	One of the supported Python feature types. This will be used to create the FC Object using addObject(). by default it is App::FeaturePython
subtype	The handler name of your ScriptedObject, by default it is the name of your class. Saved as Proxy. Type
view_proxy	A reference to the view proxy class
extensions *	A list of extensions to be added to the ScriptedObject
version	Current version of the class. (Used by migrations)
view_provider_name_override	Forced ViewProvider name

## @view\_proxy

Converts a user defined class into a full blown ViewProxy with all of the lifecycle management, proxyfied properties, extensions, display mode builders, etc...

```
1 @view_proxy(
2    view_provider_name_override: str = None,
3    extensions: Iterable[str] = None,
4    icon: str = None)
```

ARGUMENT	DESCRIPTION
view_provider_name_override	ViewProvider internal type name, empty by default so FreeCAD will decide the value
icon	Path of the icon for the Tree. If prefixed with 'self:' the path is relative to the file where the class is declared. i.e. 'self:my_icon.svg' will be resolved in the same folder as the file that declares your class.
extensions *	A list of extensions to be added to the VP

## **Display Modes**

Display modes are named zones in the 3D view that have specific presentation attributes. So objects placed in each zone are rendered with the zone's attributes.

Display modes are implemented by FreeCAD using coin objects, usually SoGroup or SoSeparator. Each display mode has a name, an optional method builder that builds the coin object and optionally can be marked as default mode.

## function: DisplayMode

```
1 def DisplayMode(name: str = None, is_default: bool = False, builder: Callable = None)
```

Declarator of Display Modes, allows to configure a mode and optionally a builder method to create and register the coin object.

ARGUMENT	ТҮРЕ	DESCRIPTION
name	str	Name of the display mode
is_default	bool	Configure the DM as default, defaults to False
builder	Callable[[ViewObject], coin.SoGroup]	Method to build the coin object if required

#### Example:

```
1 @view_proxy()
2 class MyViewProxy:
3    wireframe_plus = DisplayMode(name="WireframePlus", is_default=True)
4    shaded = DisplayMode(name="Shaded")
5    @wireframe_plus.builder
7    def wireframe_plus_builder(self, vp):
8        return SoSeparator()
```

## **Extensions**

Extensions are predefined behaviors that can be added to the DocumentObject or ViewObject to add functionality.

#### **Available Object Extensions**

- App::GeoFeatureGroupExtensionPython
- App::GroupExtensionPython
- App::LinkBaseExtensionPython
  - App::LinkExtensionPython (ref)

- App::OriginGroupExtensionPython
- App::SuppressibleExtensionPython
- Part::AttachExtensionPython
- TechDraw::CosmeticExtensionPython

#### **Available View extensions**

- Gui::ViewProviderSuppressibleExtensionPython
- Gui::ViewProviderExtensionPython
- Gui::ViewProviderGeoFeatureGroupExtensionPython
- Gui::ViewProviderGroupExtensionPython
- Gui::ViewProviderOriginGroupExtensionPython
- PartGui::ViewProviderAttachExtensionPython
- PartGui::ViewProviderSplineExtensionPython

# Migrations

Migrating old versions of your ScriptedObject to maintain backwards compatibility with old files is a complex topic. You can read all the low level details in this extensive official wiki: https://wiki.freecad.org/Scripted\_objects\_migration

In this API, migrations are way more simple as you only have to add the migrations decorator and implement the corresponding methods

### Migrations using the same DataProxy Class

```
1
2 @migrations()
3 @proxy(version=2)
   class FpoClass:
5
6
       def on_migrate_complete(self, version, obj):
           # Called after all migrations are applied
 7
8
9
       def on_migrate_upgrade(self, version, fp):
           # Called if version is less than current version
10
            # Do any required migration code here
11
12
13
       def on_migrate_downgrade(self, version, fp):
           # Called if version is greater than current version
14
            # Do any required migration code here
15
16
```

```
def on_migrate_error(self, version, fp):
    # Called if migration fails

19
20 ...
21
```

## Migrations using a different DataProxy class

Some times you refactor your code and move the file that declares your DataProxy class, in this scenario FreeCAD fails to find your class as its old module is persisted in the FCStd file. In this situation you need to do a redirection from the old file to the new one.

Suppose that your old DataProxy was defined as a class named OriginalFpo in a file named original.py and you decided to move it to a file named better.py and renamed your class to BetterFpo. You need to redirect calls from the old file/class to the new one, and also apply some migration logic to convert the old version into the new version.

In your new file better.py you have your new class, nothing special is required there.

```
# file: better.py

2

@view_proxy()

class BetterFpoViewProvider:

# ....

@proxy(view_provider=BetterFpoViewProvider)

class BetterFpo:

# All the new stuff

10
```

Now you have to redirect old calls from the old file to the new one. So just create a class with the old name but make it into a migration, the migration will take care of calling your logic and redirecting to the new class after it.

```
1 # file: original.py
   from fpo import migrations, proxy
   from better import BetterFpo, BetterFpoViewProvider
5
   @migrations(current=BetterFpo)
6
   @proxy()
   class OriginalFpo:
8
9
       def on_migrate_class(self, version, fp):
10
           # Perform any migration logic here ....
11
12
            # Then rebind to the new class
13
           BetterFpo.rebind(fp) # Reinitialize fp as the new Fpo
14
15
```

# @migrations decorator

# function: migrations

```
1 def migrations(current=None)
```

Install migrations management into the class.

ARGUMENT	ТҮРЕ	DESCRIPTION
current	ProxyClass	most recent class, if omitted, the same class is used.

#### Example:

```
1 @migrations()
2 @proxy(version=5)
3 class MyScriptedObjectClass:
4 ...
```

## FreeCAD Preferences

You can read and write FreeCAD preferences from your code using a simple API. Use it to save/load configurations.

In FreeCAD, preferences are saved in a Tree of groups/entries like this:

- Every Group is under a root parent, in the example above the root is BaseApp
- Every Entry is under a Group, in the example above the group is MyExtension/My Group
- Every Entry has one value of type int, float, str, bool

To access them for read/write, you just need to create a proxy for the preference:

```
2 #----
3 # file: preferences.py
4 # Declare preferences wherever you want,
5 # but usually in some `preferences.py` module
6 # so you can reuse from everywhere.
7 from fpo import Preference
8 config_x = Preference(group="MyExtension/My Group", name="My Param X", default=10)
9 config_y = Preference(group="MyExtension/My Group", name="My Param Y", default=10)
10
   config_z = Preference(group="MyExtension/My Group", name="My Param Z", default=10)
11
12 #----
13 # file: whatever.py
14 import preferences as pref
15
16 # read values
17 print(f"X = {pref.config_x()}")
18 print(f"Y = {pref.config_y()}")
19
   print(f"Z = {pref.config_z()}")
21 # write values
22 pref.config_x(150)
```

```
pref.config_y(100)
pref.config_z(210)

# subscribe/observe to changes in preferences with listeners
from fpo import Preference

@Preference.subscribe(group="MyExtension/My Group")
def on_preference_change(group, value_type, name, value):
    print(f"Preference changed: {group}, {value_type}, {name}, {value}")
```

## **Preferences API**

#### Preference

```
1 Preference(group:str, name:str, default:Any=None, value_type:type=str, root:str="BaseApp")
```

ARGUMENT	DESCRIPTION
group	Group path
name	Entry name
default	Default value returned if Entry does not exists
value_type	Type of the value, if not provided, type(default) is used, if no default. str is used
root	Tree root, default is BaseApp

## @Preference.subscribe

```
1 @Preference.subscribe(group:str, root="BaseApp")
```

Creates and attach a preference listener, you can observe changes in a group.

ARGUMENT	DESCRIPTION
group	Group path to observe
root	Tree root, default is BaseApp

```
from fpo import Preference

Preference.subscribe(group="MyExtension/My Group")

def on_preference_change(group, value_type, name, value):
    print(f"Preference changed: {group}, {value_type}, {name}, {value}")

# You can remove the observer subscription later:
    on_preference_change.unsubscribe()
```

# Utility functions reference

There are also few global functions that are frequently used in ScriptedObject development. So I included them here for quick reference because they are used in the examples.

## Global functions in fpo module

## function: get\_selection

```
1 def get_selection(*args) -> Tuple
```

Returns current selection in specific order and matching specific types.

case 1: no args, just returns selection as list:

```
1 sel = get_selection()
```

case 2: args are the required selection types:

```
1 ok, axis, obj = get_selection('PartDesign::Line'. '*')
2 if ok:
3 ...
```

regex are supported for type matching and '\*' is a general wildcard:

```
ok, axis, part, other = get_selection('PartDesign::Line', re.compile('Part::.*'), '*')
if ok:
...
```

this will parse selection for three elements, first one must be a PartDesign::Line, second one must be any object from the Part namespace, the last one can be anything. The user can select them in any order but they will be returned in the specified order. Take into account that wildcards will match anything so it is better to specify patterns from more specific to least specific.

In this invocation schema, the first element of the returned tuple is a boolean that indicate if the selection matches the patterns.

RETURN TYPE	DESCRIPTION
List[DocumentObject]	If no arguments are supplied, the list of selected objects
bool, *List[DocumentObject]	If arguments are supplied, the first element returned says if the selection matches the patterns, the rest are the selected objects in order

ARGUMENT	ТҮРЕ	DESCRIPTION
*args	*[str re.Pattern]	List of patterns

## function: set\_immutable\_prop

```
1 def set_immutable_prop(obj: ObjectRef, name: str, value: Any) -> None
```

Force update a property with Immutable status. It temporarily removes the immutable flag, sets the value and restore the flag if required.

ARGUMENT	ТҮРЕ	DESCRIPTION
obj	ObjectRef	remote FreeCAD object
name	str	property
value	Any	the value

## function: message\_box

```
1 def message_box(message: str, title: str = 'Message', details: str = None)
```

Shows a basic message dialog (modal) if App. GuiUp is True, else prints to the console.

ARGUMENT	ТҮРЕ	DESCRIPTION
message	str	summary
title	str	box title, defaults to "Message"
details	str	expandable text, defaults to None

## function: confirm\_box

```
1 def confirm_box(message: str, title: str = 'Message', details: str = None) -> bool
```

Ask for a confirmation with a basic dialog. Requires App.GuiUp == True

RETURN TYPE	DESCRIPTION
bool	True if user accepts

ARGUMENT	ТҮРЕ	DESCRIPTION
message	str	summary
title	str	box title, defaults to "Message"
details	str	expandable text, defaults to None

## function: print\_log

```
1 def print_log(*args)
```

Print into the FreeCAD console with info level.

## function: print\_err

```
1 def print_err(*args)
```

Print into the FreeCAD console with error level.

## function: get\_pd\_active\_body

```
1 def get_pd_active_body() -> PartDesign_Body
```

Retrieve the active PartDesign Body if any

RETURN TYPE	DESCRIPTION
PartDesign_Body	Active Body

# function: set\_pd\_shape

```
1 def set_pd_shape(fp: DocumentObject, shape: Shape) -> None
```

Prepare the shape for usage in PartDesign and sets BaseFeature and AddSubShape

## Compatibility notes

#### Serialization and deserialization

State serialization process used to be managed by methods named \_\_getstate\_\_ / \_\_setstate\_\_ in older versions of FreeCAD but they were renamed to dumps / loads in recent versions due to conflicts with python 3.11+.

This backwards compatibility issue is transparently managed by this API, but it also was fixed in master recently:

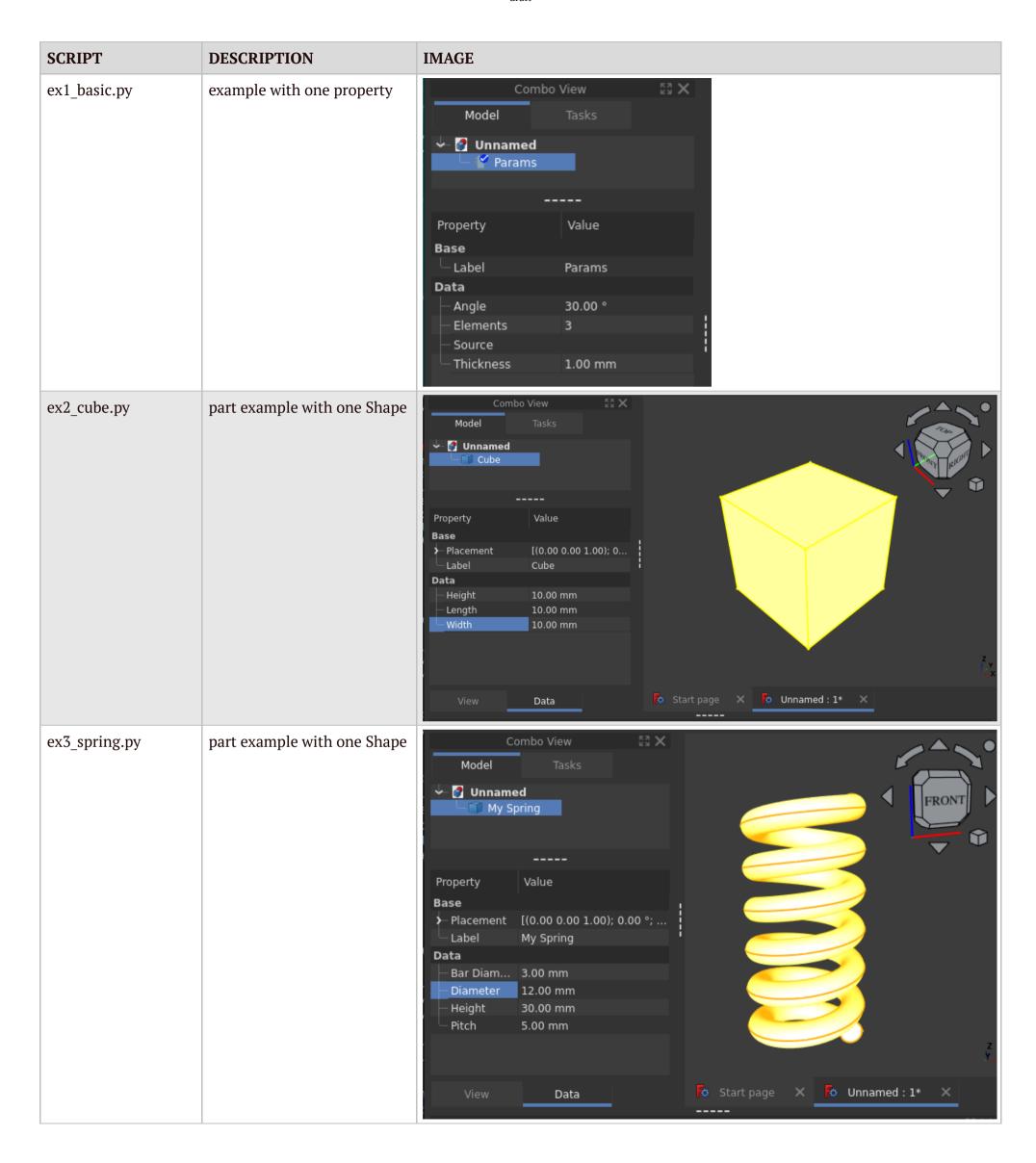
- https://github.com/FreeCAD/FreeCAD/pull/12243
- https://github.com/FreeCAD/FreeCAD/pull/10769

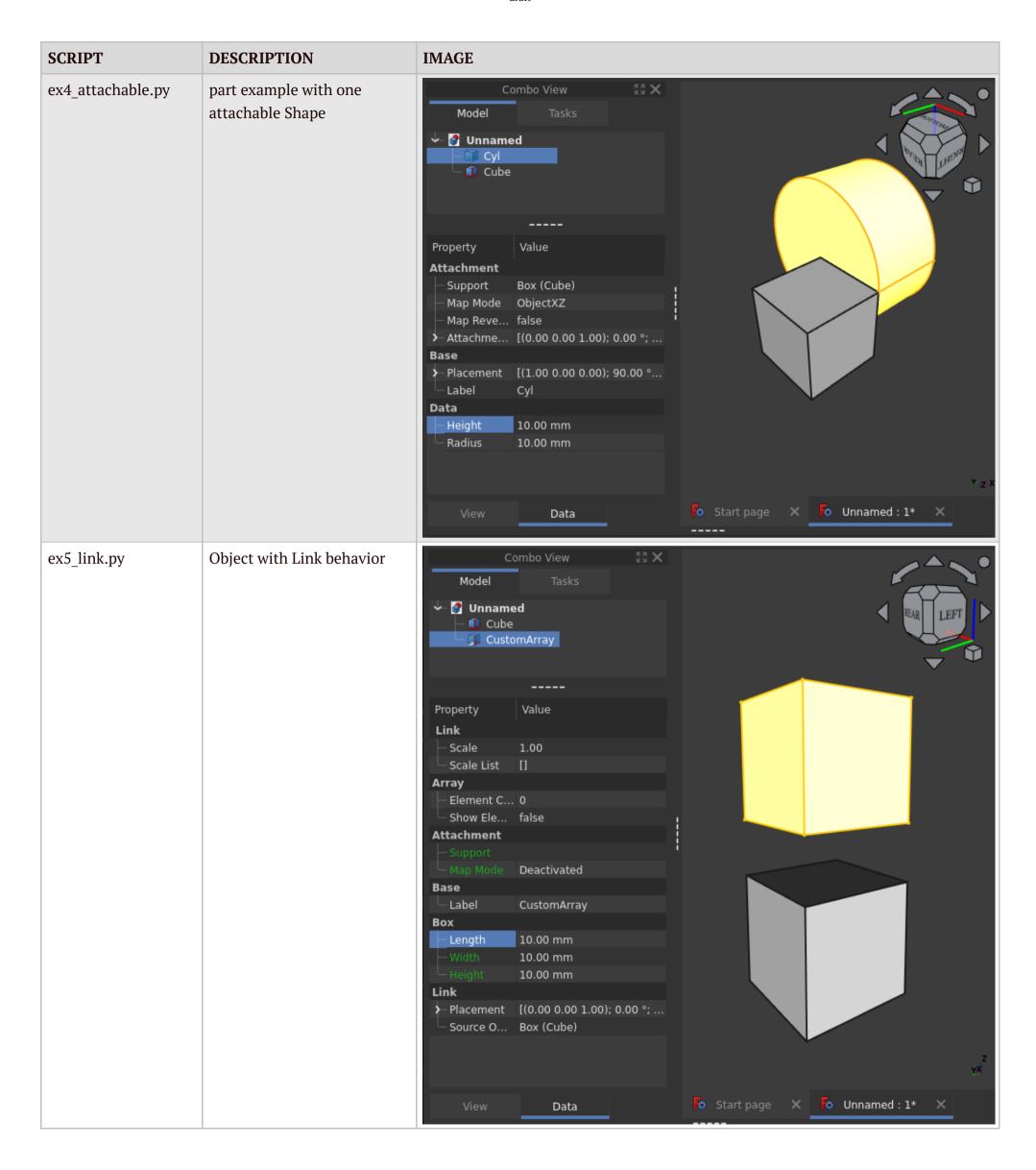
#### Official documentation sources:

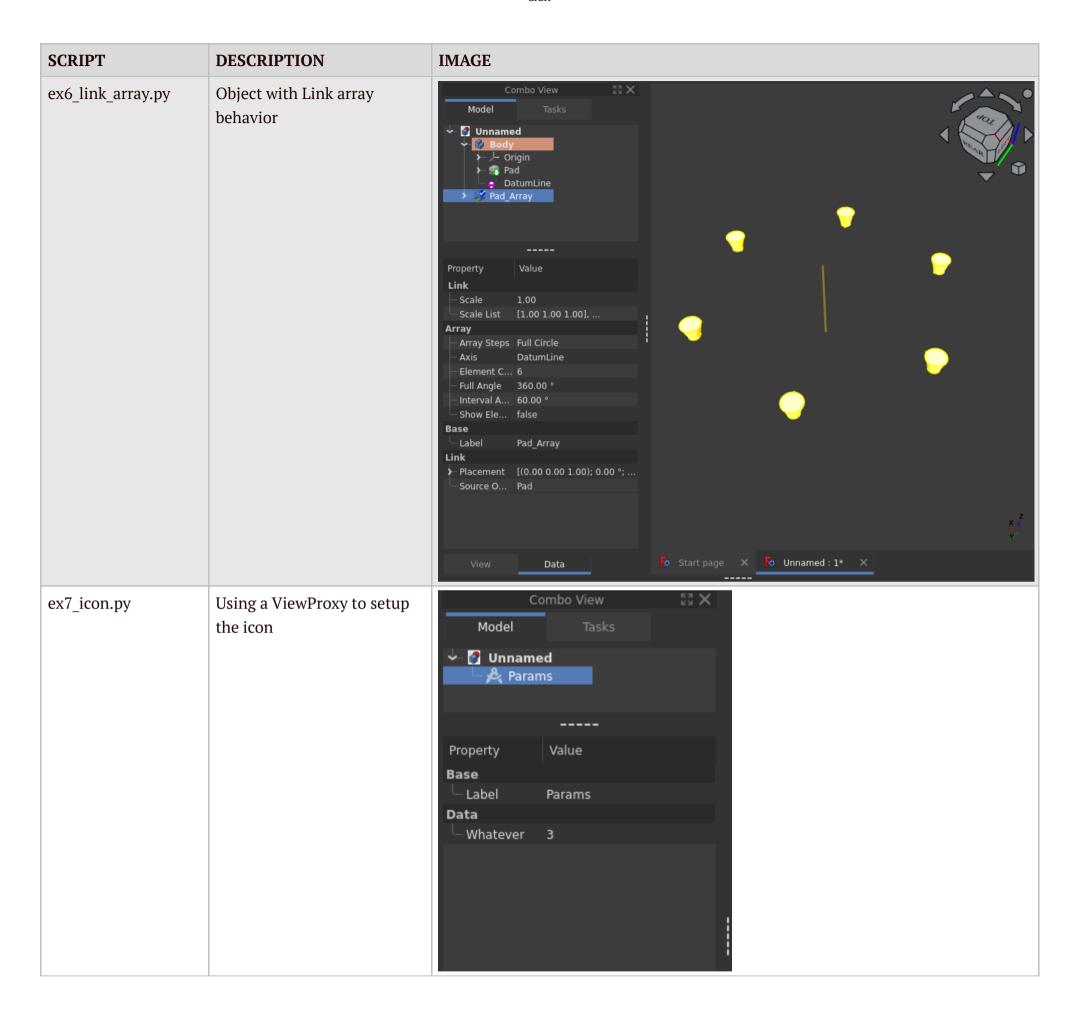
- https://wiki.freecad.org/App FeaturePython
- https://wiki.freecad.org/Viewprovider
- https://wiki.freecad.org/FeaturePython\_Custom\_Properties
- https://wiki.freecad.org/Create a FeaturePython object part I
- https://wiki.freecad.org/Create a FeaturePython object part II
- https://wiki.freecad.org/Scripted objects
- https://wiki.freecad.org/Scripted objects migration
- https://forum.freecad.org/viewforum.php?f=22
- https://wiki.freecad.org/File:FreeCAD core objects.svg

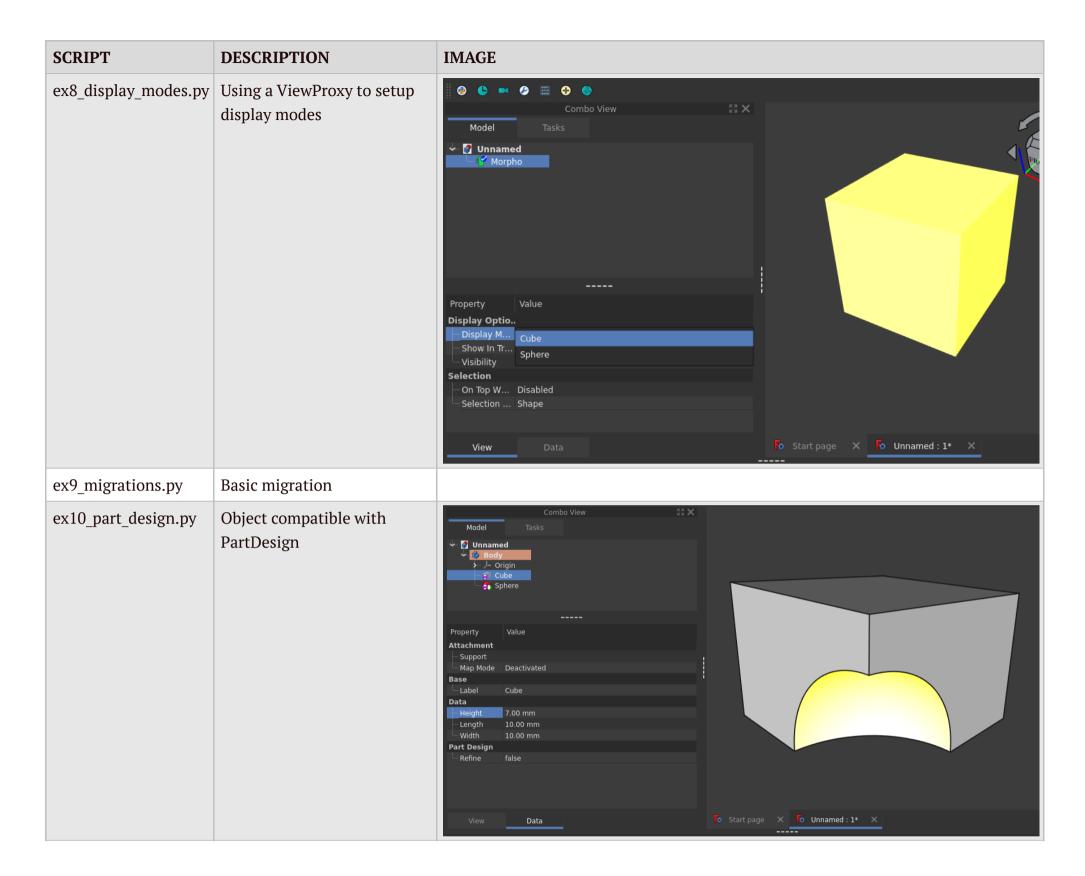
# Code examples

There are some basic examples in the examples folder. The examples are numbered in order to imply increasing complexity, I do not repeat comments that were already present in previous examples.









# Quick setup

The only required file to use this API is fpo.py, the key is where to put them as it is not a FreeCAD core thing by now.

## Usage from a Workbench

Put fpo.py in your *Workbench* folder and use it. It is supposed that this API is used for *Workbench* developers so no other special configuration is required for that case.

As fpo.py is not part of the core FreeCAD distribution, it is possible that other *Workbenches* already include the file. So it is a good precaution to rename your copy or put it in your internal module.

Remember that the recommended layout for workbenches is to put the code into a sub-module of freecad.

```
1
   └─ FreeCAD Config Dir/
       └─ Mod/
3
           └─ YourWorkbench/
               └─ freecad/
5
                   └─ your_module/
6
                       __init__.py
8
                       ├─ init_gui.py
                       ├─ fpo.py
                       your_amazing_thing.py
10
11
```

#### **Usage from Macros**

It is better to not define ScriptedObject classes directly in **Macros** because FreeCAD will have have a hard time finding them when reloading the objects from saved documents.

What you can do in this case is putting the fpo.py file directly in the FreeCAD's **Macros** directory, then create your ScriptedObject classes in its own file in **Macros** dir, then import them from your macros. That way FreeCAD will find the Proxies next time you open your Documents.

### Quick and dirty setup

Another easy way if you don't want to develop a *Workbench* is to fake one, To do that simply create a folder inside FreeCAD's Mod directory and put fpo.py and your other python files there along with an empty \_\_init\_\_.py file. This will make fpo and your modules visible to FreeCAD in the main path.

#### **Examples setup**

Copy fpo.py and examples/\* into FreeCAD's *Macro* dir. then you can run the examples from the FreeCAD's python console:

```
import ex3_spring as ex3
ex3.create_spring()

import ex10_part_design as ex10
ex10.create_cube_pd()

...
```



Copy fpo.py in only one place to avoid a name conflicts.