FreeCAD Scripted Objects Modern API

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author	Frank David Martínez Muñoz
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contributors	Gaël Écorchard

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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, coding style and limited understanding of FreeCAD internals. 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.

General programming experience, some basic FreeCAD know-how and a minimalistic comprehension of Python are sufficient, as long as you can search the internet for a basic grasp of 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 0.21+
- The API must be an overlay on top of the existing API, so it must not conflict 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.

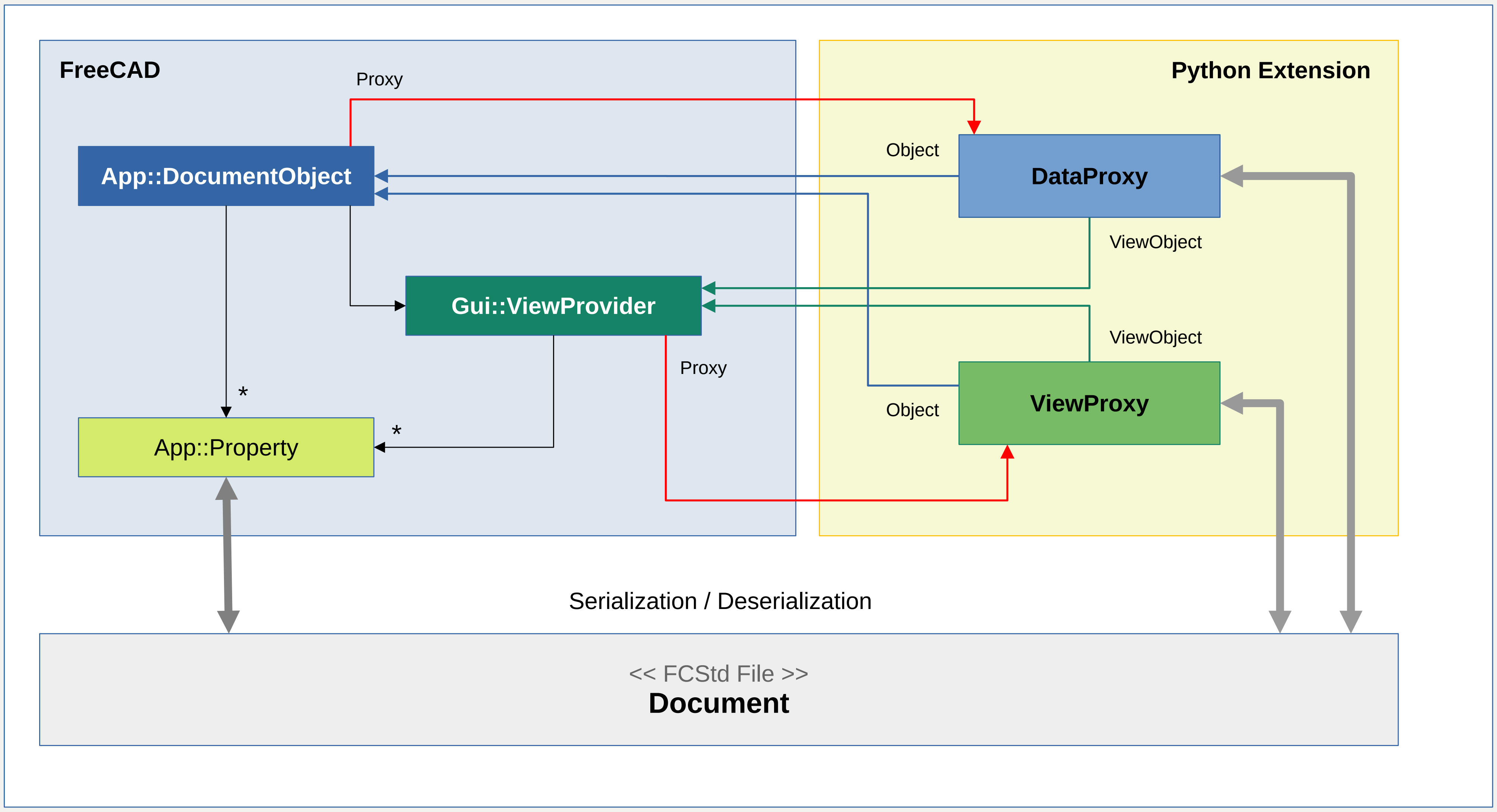
Features

- √ Declarative DataProxy (@proxy)
    - √ Observable explicit and well defined lifecycle
    - √ Serialization/Deserialization
    - √ Automatic object creation and Proxy-Object association
    - √ Declarative extensions
    - √ Extension lifecycle management
- √ Properties
    - √ Declarative creation (Property*)
    - √ Proxy based read/write
    - √ Observable
- √ Declarative ViewProxy (@view_proxy)
    - √ Display Modes
        - √ Declarative creation (DisplayMode)
        - √ Builder based creation of display modes
    - √ Drag and Drop support
- √ Migrations (@migrations)
    - √ Declarative support
    - √ Redirect to different class
    - √ Automatic version management
    - √ upgrade/downgrade/redirect
- √ Extensions
    - √ Declarative Extension support
- √ GUI utilities (`fcgui`)
- √ Preferences (`fpo.Preference`)
    - √ Proxy based read/write
    - √ Declarative listeners (@Preference.subscribe)
    - √ Automatic type handling
- √ Documentation in markdown format.

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



Important
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`.

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:

OBJECT TYPE	DESCRIPTION
`App::DocumentObjectGroupPython`	
`App::FeaturePython`	Typical Scripted Object
`App::GeometryPython`	
`App::LinkElementPython`	
`App::LinkGroupPython`	
`App::LinkPython`	
`App::MaterialObjectPython`	
`App::PlacementPython`	
`Fem::ConstraintPython`	
`Fem::FeaturePython`	
`Fem::FemAnalysisPython`	
`Fem::FemMeshObjectPython`	
`Fem::FemResultObjectPython`	
`Fem::FemSolverObjectPython`	
`Mesh::FeaturePython`	
`Part::CustomFeaturePython`	
`Part::FeaturePython`	Typical Scripted object with Shape
`Part::Part2DObjectPython`	
`PartDesign::FeatureAddSubPython`	Additive/Subtractive PD Shape
`PartDesign::FeatureAdditivePython`	Additive PD Shape
`PartDesign::FeaturePython`	Base PD Feature
`PartDesign::FeatureSubtractivePython`	Subtractive PD Shape
`PartDesign::SubShapeBinderPython`	
`Path::FeatureAreaPython`	
`Path::FeatureAreaViewPython`	
`Path::FeatureCompoundPython`	
`Path::FeaturePython`	

OBJECT TYPE	DESCRIPTION
`Path::FeatureShapePython`	
`Points::FeaturePython`	
`Sketcher::SketchObjectPython`	
`Spreadsheet::SheetPython`	
`TechDraw::DrawComplexSectionPython`	
`TechDraw::DrawLeaderLinePython`	
`TechDraw::DrawPagePython`	
`TechDraw::DrawRichAnnoPython`	
`TechDraw::DrawTemplatePython`	
`TechDraw::DrawTilePython`	
`TechDraw::DrawTileWeldPython`	
`TechDraw::DrawViewPartPython`	
`TechDraw::DrawViewPython`	
`TechDraw::DrawViewSectionPython`	
`TechDraw::DrawViewSymbolPython`	
`TechDraw::DrawWeldSymbolPython`	

- Wiki Source: https://wiki.freecad.org/Scripted_objects#Available_object_types
- Forum Source: https://forum.freecad.org/viewtopic.php?t=86414&start=10#p752318

> **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. The parametric effect of the `ScriptedObject` is not achieved by inheritance but rather by "parallel collaboration". The FreeCAD object has a member, `Proxy`, that is the instance of the `DataProxy` class that was created by passing the object itself as argument. Properties belong to the object and trigger callbacks in the proxy, if any. This is the main way scripted objects work. There exist also FreeCAD generated events that trigger callbacks in the proxy.

This class is also responsible for serializing/deserializing its own internal state (i.e. not object's properties) from/to the document.

Inside the proxy methods, the object is accessible via `self.Object`.

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

```python
from fpo import proxy, PropertyLength, print_log

@proxy()
class MyCustomObjectProxy:
    length = PropertyLength(default=5)

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

# -- usage
obj = 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. Analogously to the `DataProxy`, the `ViewProxy` is not inherited but rather works in parallel with the `ViewObject` (accessible via `object.ViewObject`). Changes in properties of the `ViewObject` and FreeCAD GUI events trigger callbacks in the `ViewProxy`. Moreover, there is a callback (`on_object_change`) for changes in the `Object` itself.

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

Inside the proxy methods, the object and the view provider are accessible via `self.Object` and `self.ViewObject`, respectively.

To define a `ViewProxy` class just define a class and annotate it with @view_proxy decorator.

```python
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)

```

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.

```python
from fpo import proxy, view_proxy

@view_proxy()
class MyCustomViewProxy:
    ...

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

# -----
obj = MyCustomObjectProxy.create(name="MyThing")
```

Object creation and binding

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

```python
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	TYPE	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:

```python
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.

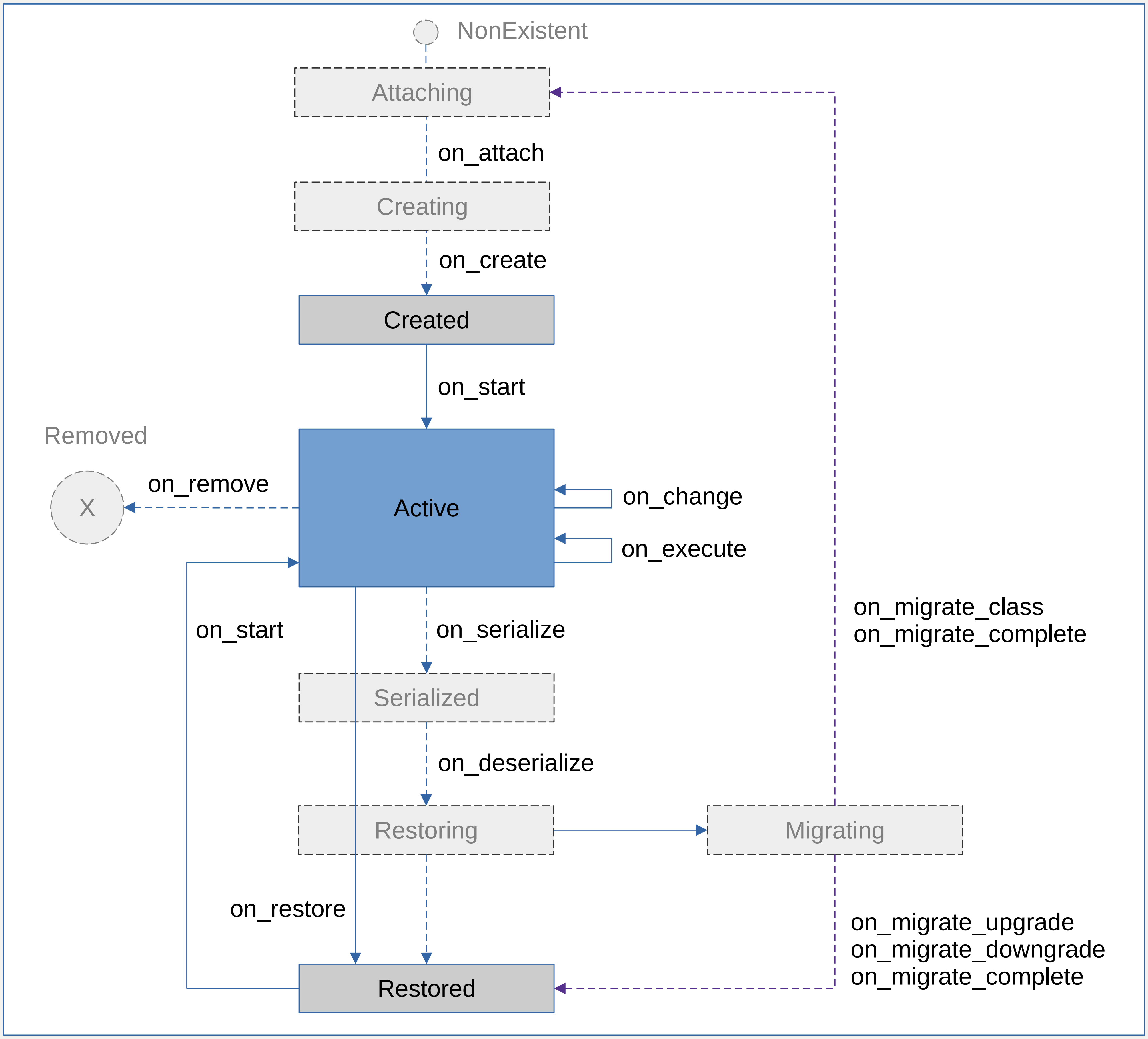
All possible states

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:

```python
from fpo import proxy

@proxy()
class MyCustomObjectProxy:

    def on_create(self, event: fpo.events.CreateEvent) -> None:
        print(f"{event.source.Label} ({event.source.Name}) was created")

    def on_attach(self, event: fpo.events.AttachEvent) -> None:
        print(f"{event.source.Label} was attached to the document")

    def on_start(self, event: fpo.events.StartEvent) -> None:
        print(f"{event.source.Label} is ready")

    def on_remove(self, event: fpo.events.RemoveEvent) -> None:
        print(f"{event.source.Name} was removed")

    def on_restore(self, event: fpo.events.DocumentRestoredEvent) -> None:
        print(f"{event.source.Name} ({event.source.Name}) was loaded from the file")

    ...
```

on_attach

```python
def on_attach(self: Proxy) -> None
def on_attach(self: Proxy, event: fpo.events.AttachEvent) -> None
```

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

```python
class AttachEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject | None = None
```

on_create

```python
def on_create(self: Proxy) -> None
def on_create(self: Proxy, event: fpo.events.CreateEvent) -> None
```

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

```python
class CreateEvent:
    source: DocumentObject
```

on_start

```
def on_start(self: Proxy) -> None
def on_start(self: Proxy, event: fpo.events.StartEvent) -> 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.

```
class StartEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject | None = None
```

on_remove

```
def on_remove(self: Proxy) -> None
def on_remove(self: Proxy, event: fpo.events.RemoveEvent) -> None
```

Called before the object is removed from the Document

```
class RemoveEvent:
    source: DocumentObject
```

on_restore

```
def on_restore(self: Proxy) -> None
def on_restore(self: Proxy, event: fpo.events.DocumentRestoredEvent) -> None
```

Called when the object is restored (read) from the *FCStd* file

```
class DocumentRestoredEvent:
    source: DocumentObject
```

Persistence events listeners

FreeCAD is responsible for managing the persistence of the `DocumentObjects` and `ViewProviders` (that means objects' properties) but your `Proxy` classes are responsible for persisting/loading its own internal state from/to the document.

on_serialize

```
def on_serialize(self: Proxy) -> None
def on_serialize(self: Proxy, event: fpo.events.SerializeEvent) -> 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.

```python
class SerializeEvent:
    source: DocumentObject
    state: dict[str, Any]
    view_provider: ViewProviderDocumentObject | None = None
```

```python
from fpo import proxy

@proxy()
class MyCustomObjectProxy:
    var1: str
    var2: int

    def __init__(self, fp):
        self.var1 = 'Hello'
        self.var2 = 5

    def on_serialize(self, event: fpo.events.SerializeEvent) -> None:
        event.state['my_value_1'] = self.var1
        event.state['my_value_1'] = self.var2

    ...
```

on_deserialize

```python
def on_deserialize(self: Proxy) -> None
def on_deserialize(self, event: fpo.events.DeserializeEvent) -> 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.

```python
class DeserializeEvent:
    source: DocumentObject
    state: dict[str, Any]
    view_provider: ViewProviderDocumentObject | None = None
```

```python
from fpo import proxy

@proxy()
class MyCustomObjectProxy:
    var1: str
    var2: int

    def on_deserialize(self, event: fpo.events.DeserializeEvent) -> None:
        self.var1 = events.state.get('my_value_1', '')
        self.var2 = events.state.get('my_value_2', 0)

    ...
```

Active event handlers

When your `ScriptedObject` is active, all your work is performed in `on_execute` and `on_change` event listeners.

To query the state of the object, you can call the `is_active` method:

```python
def is_active(self: Proxy) -> bool
```

You can also override the method to provide your own logic.

on_execute

```python
def on_execute(self: Proxy) -> None
def on_execute(self: Proxy, event: events.ExecuteEvent) -> 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).

```python
class ExecuteEvent:
    source: DocumentObject
```

```python
from fpo import proxy, PropertyLength
import Part

@proxy(object_type='Part::FeaturePython')
class CustomBoxProxy:
    width = PropertyLength(default=5.0, description='Width of the box')
    length = PropertyLength(default=5.0, description='Length of the box')
    height = PropertyLength(default=5.0, description='Height of the box')

    def on_execute(self):
        # Your magic happens here
        self.Object.Shape = Part.makeBox(self.length, self.width, self.height)

    ...

# -----
obj = CustomBoxProxy.create(name='box1')
```

on_change

```python
def on_change(self: Proxy, event: events.PropertyChangedEvent) -> None
```

Called after any property has changed. Note that you would normally not implement this override but rather use listeners described below.

```python
class PropertyChangedEvent(Generic[PT]):
    source: DocumentObject
    property_name: str
    old_value: events.PT | None
    new_value: events.PT | None
    view_provider: ViewProviderDocumentObject | None = None
```

on_before_change

```python
def on_before_change(self: Proxy) -> None
def on_before_change(self: Proxy, event: events.PropertyWillChangeEvent) -> None
```

Called before a property change is performed.

```python
class PropertyWillChangeEvent(Generic[PT]):
    source: DocumentObject
    property_name: str
    value: events.PT | None
    view_provider: ViewProviderDocumentObject | None = None
```

Direct property change listeners

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

```python
@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')
    height = PropertyFloat(default=5.0, description='Height of the box')

    # Your magic happens here
    def on_execute(self):
        self.Object.Shape = Part.makeBox(self.length, self.width, self.height)

    @length.observer
    def length_changed(self, event: events.PropertyChangedEvent) -> None:
        print(f"Hey! length has changed from {event.old_Value} to {event.new_value}")

```

They are compatible with the `on_change` event handler, so you can use both. `on_change()` is called before or after calling the property listener.

Other listeners

on_extension

```python
def on_extension(self: Proxy, event: events.ExtensionEvent) -> 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.

```python
class ExtensionEvent:
    source: DocumentObject
    name: str
```

Hooks

There are other optional methods called by FreeCAD to get some info from the Proxy.

is_active

```python
def is_active(self) -> bool
```

Return True if your `DataProxy` in the "ready" state.

is_dirty

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

Return True if your `DataProxy` 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:

```
@proxy()
class MyMagicProxy:
    my_property = PropertyInteger(section="Basic", default=5)

    # Optional listener
    @my_property.observer
    def my_property_obs(self, event: events.PropertyChangedEvent):
        print(f"MyProperty has changed from {event.old_value} to {event.new_value}")

```

Property constructors

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

ARGUMENT	DESCRIPTION
name	Name of the property, deduced from the attribute if missing

ARGUMENT	DESCRIPTION
section	Subsection 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. Signature `function(event: events.PropertyChangedEvent)`

Examples

```python
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

A function can be subscribed to the property to listen for change events. The argument of the property listener is optional

```
@proxy()
class MyMagicProxy
    my_prop1 = PropertyInteger(section="Basic", default=5)
    my_prop2 = PropertyInteger(section="Basic", default=5)

    @my_prop1.observer
    def listener1(self, event: events.PropertyChangedEvent):
        print(f"MyProperty1 has changed from {event.old_value} to {event.new_value}")

    @my_prop2.observer
    def listener2(self):
        print("MyProperty2 has changed")

```

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)

    def on_execute(self):

        # Transparently access the property from the remote object
        # The returned value is a float when the property is a quantity
        # (length, angle, etc...).
        x = self.my_property1

        # You can also access the property from the object.
        # Remember that the name is automatically camel-cased if not specified
        # in the constructor.
        # This time, xx is an `App.Units.Quantity` where applicable.
        xx = self.Object.MyProperty1

        # Transparently update the property from the remote object
        self.my_property1 = 10

```

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 original FreeCAD API, but in that case you manage them directly from the object.

```
@proxy()
class MyMagicProxy

    def on_start(self) -> None:
        if not hasattr(self.Object, 'Length'):
            self.Object.addProperty(
                "App::PropertyLength",
                "Length", "Box", "Length of the box").Length = 1.0

    def on_execute(self, event: events.ExecuteEvent) -> None:
        # read (a Quantity here)
        x = self.Object.Length
        # write
        self.Object.Length = 10

```

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

```
def on_attach(self) -> None
def on_attach(self, event: events.AttachEvent) -> None
```

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

on_start

```python
def on_start(self) -> None
def on_start(self, event: events.StartEvent) -> None
```

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

on_edit_start

```python
def on_edit_start(self, event: events.EditStartEvent) -> bool | None
```

Called when the user requests edit. See edit modes

```python
class EditStartEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    mode: int
```

on_edit_end

```python
def on_edit_end(self, event: events.EditEndEvent) -> bool | None
```

Called when the user terminates editing. See edit modes

```python
class EditEndEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    mode: int
```

on_dbl_click

```python
def on_dbl_click(self, event: events.DoubleClickEvent) -> bool
```

Called when the user double clicks the Tree Node. Return True to tell the core system that you handled the action already.

```python
class DoubleClickEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
```

on_context_menu

```
def on_context_menu(self, event: events.ContextMenuEvent) -> None
```

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

```
class ContextMenuEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    menu: QtGui.QMenu
```

```
def on_context_menu(self, event: events.ContextMenuEvent) -> None:
    event.menu.addAction(...)
```

on_delete

```
def on_delete(self, event: events.DeleteEvent) -> bool
```

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

```
class DeleteEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    sub_elements: Any
```

on_claim_children

```
def on_claim_children(self, event: events.ClaimChildrenEvent) -> list[DocumentObject]
```

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

```
class ClaimChildrenEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
```

on_drag_object

```
def on_drag_object(self, event: events.DragAndDropEvent) -> None
```

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

```python
class DragAndDropEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    dragged_object: DocumentObject
```

on_drop_object

```python
def on_drop_object(self, event: events.DragAndDropEvent) -> None
```

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

on_object_change

```python
def on_object_change(self, event: events.DataChangedEvent) -> None
```

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

```python
class DataChangedEvent:
    source: DocumentObject
    view_provider: ViewProviderDocumentObject
    property_name: str
```

ViewProxy hooks

---

can_drag_objects

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

Returns True if this VP accepts dragging of sub-elements.

can_drop_objects

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

Returns True if this VP accepts dropping of sub-elements.

can_drag_object

```python
def can_drag_object(self, event: events.DragAndDropEvent) -> bool
```

Returns True if this VP accepts dragging of the dragged `obj`.

can_drop_object

```python
def can_drop_object(self, event: events.DragAndDropEvent) -> bool
```

Returns True if this VP accepts dropping of the incoming `obj`.

icon

```python
def icon(self) -> str | None
```

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

```python
@proxy(
    object_type: str = 'App::FeaturePython',
    subtype: str = None,
    view_proxy: ViewProxy = None,
    extensions: Iterable[str] = None,
    view_provider_name_override: str = None,
    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...

```python
@view_proxy(
    view_provider_name_override: str = None,
    extensions: Iterable[str] = None,
    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

Signature / DisplayMode

```python
def DisplayMode(
        name: str | None=None,
        *,
        is_default: bool=False,
        builder: Callable | None=None
    ) -> None: ...
```

Docs / DisplayMode

ARGUMENT	TYPE	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

Declare a display mode.

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

Example:

```python
@view_proxy()
class MyViewProxy:
    wireframe_plus = DisplayMode(name="WireframePlus", is_default=True)
    shaded = DisplayMode(name="Shaded")

    @wireframe_plus.builder
    def wireframe_plus_builder(self, vp: "ViewObject"):
        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

```

@migrations()
@proxy(version=2)
class FpoClass:

    def on_migrate_complete(self, event: events.MigrationEvent) -> None:
        # Called after all migrations are applied

    def on_migrate_upgrade(self, event: events.MigrationEvent) -> None:
        # Called if version is less than current version
        # Do any required migration code here

    def on_migrate_downgrade(self, event: events.MigrationEvent) -> None:
        # Called if version is greater than current version
        # Do any required migration code here

    def on_migrate_error(self, event: events.MigrationEvent) -> None:
        # Called if migration fails

    ...

```

```
class MigrationEvent:
    source: DocumentObject
    from_version: int
    to_version: int
```

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.

```python
# file: better.py

@view_proxy()
class BetterFpoViewProvider:
    # ....

@proxy(view_provider=BetterFpoViewProvider)
class BetterFpo:
    # All the new stuff

```

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.

```python
# file: original.py

from fpo import migrations, proxy
from better import BetterFpo, BetterFpoViewProvider

@migrations(current=BetterFpo)
@proxy()
class OriginalFpo:

    def on_migrate_class(self, event: events.MigrationEvent) -> None:
        # Perform any migration logic here ....

        # Then rebind to the new class
        BetterFpo.rebind(fp) # Reinitialize fp as the new Fpo

```

@migrations decorator

Function: migrations

Signature / migrations

```python
def migrations(current: type | None=None) -> Callable[[type], type]: ...
```

Docs / migrations

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

Install migrations management into the class.

Example:

```
@migrations()
@proxy(version=5)
class MyScriptedObjectClass:
    ...
```

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:

```
.
└── User parameter:BaseApp/
    └── MyExtension/
        └── My Group/
            ├── My Param X = 0.5
            ├── My Param Y = 10
            └── My Param Z = 100
```

- 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:

```python

#------
# file: preferences.py
#  Declare preferences wherever you want,
#  but usually in some `preferences.py` module
#  so you can reuse from everywhere.
#  If you do not use `default`, you must use `value_type` (e.g. `value_type=int`)
#  otherwise the default type is `str`.
#  Beware, that there can exist multiple parameters in the same group and with the
#  same name if they have different types, so setting the type is important.
from fpo import Preference
config_x = Preference(group="MyExtension/My Group", name="My Param X", default=10)
config_y = Preference(group="MyExtension/My Group", name="My Param Y", default=10)
config_z = Preference(group="MyExtension/My Group", name="My Param Z", default=10)

#------
# file: whatever.py
import preferences as pref

# read values
print(f"X = {pref.config_x()}")
print(f"Y = {pref.config_y()}")
print(f"Z = {pref.config_z()}")
```

```python

# write values
pref.config_x(update=150)
pref.config_y(update=100)
pref.config_z(update=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

```python
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

```python
@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

```python
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 include them here for quick reference because they are used in the examples.

Global functions in fpo module

Function: get_selection

Signature / get_selection

```python
def get_selection(*args: tuple) -> tuple: ...
```

Docs / get_selection

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	TYPE	DESCRIPTION
*args	*[str|re.Pattern]	List of patterns

Return current selection in specific order and matching specific types.

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

```python
sel = get_selection()
```

case 2: args are the required selection types:

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

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

```python
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.

Function: set_immutable_prop

Signature / set_immutable_prop

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

Docs / set_immutable_prop

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

Force update a property with Immutable status.

Temporarily removes the immutable flag, sets the value and restore the flag if required.

Function: message_box

Signature / message_box

```python
def message_box(message: str, title: str='Message', details: str='') -> None: ...
```

Docs / message_box

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

Show a basic message dialog (modal).

If `App.GuiUp` is False, prints to the console.

Function: confirm_box

Signature / confirm_box

```python
def confirm_box(message: str, title: str='Message', details: str='') -> bool: ...
```

Docs / confirm_box

RETURN TYPE	DESCRIPTION
bool	True if user accepts

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

Ask for a confirmation with a basic dialog.

Requires App.GuiUp == True

Function: print_log

Signature / print_log

```python
def print_log(*args: tuple) -> None: ...
```

Docs / print_log

Print into the FreeCAD console with info level.

Function: print_err

Signature / print_err

```python
def print_err(*args: tuple) -> None: ...
```

Docs / print_err

Print into the FreeCAD console with error level.

Function: get_pd_active_body

Signature / get_pd_active_body

```python
def get_pd_active_body() -> DocumentObject | tuple[DocumentObject, DocumentObject, str] | None: ...
```

Docs / get_pd_active_body

RETURN TYPE	DESCRIPTION
PartDesign.Body	Active Body

Retrieve the active PartDesign Body if any.

Function: set_pd_shape

Signature / set_pd_shape

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

Docs / set_pd_shape

Prepare the shape for usage in PartDesign and sets `Shape` 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

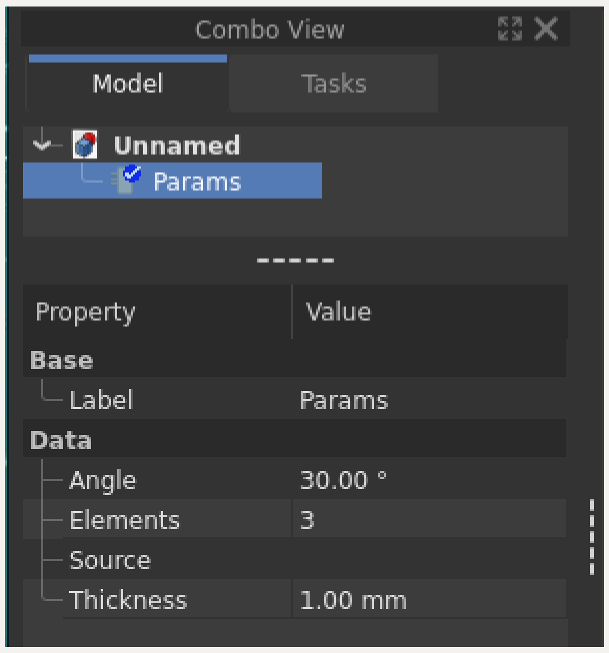
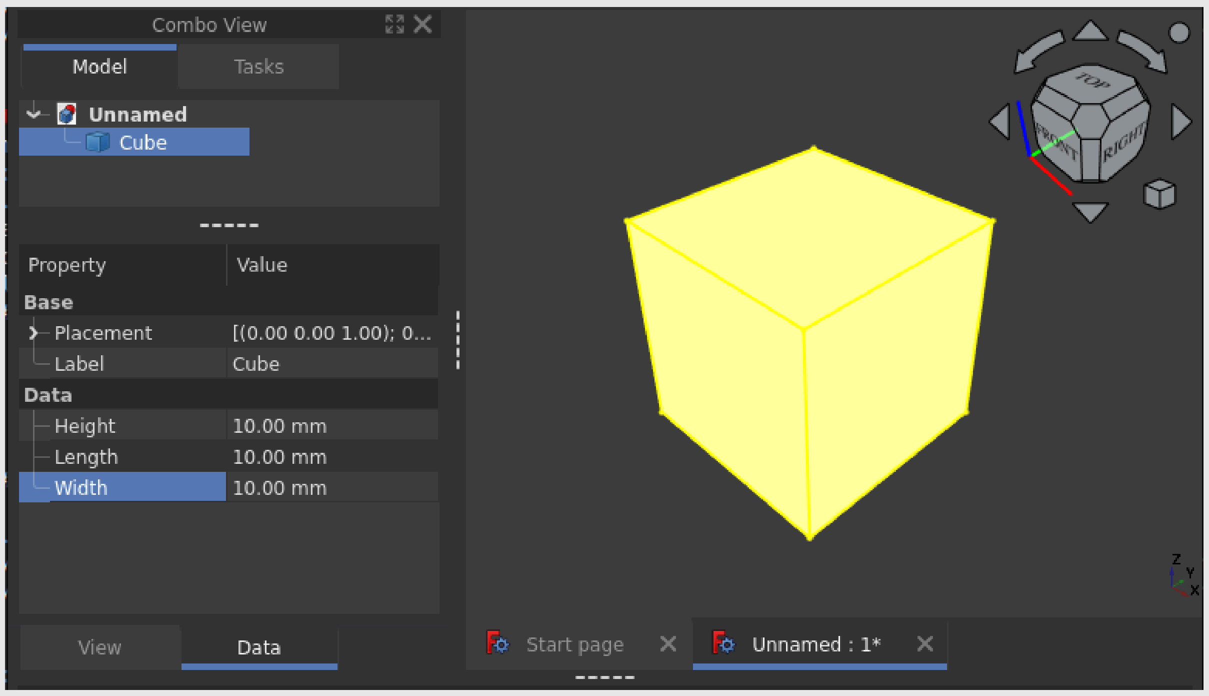
Official documentation sources:

- 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

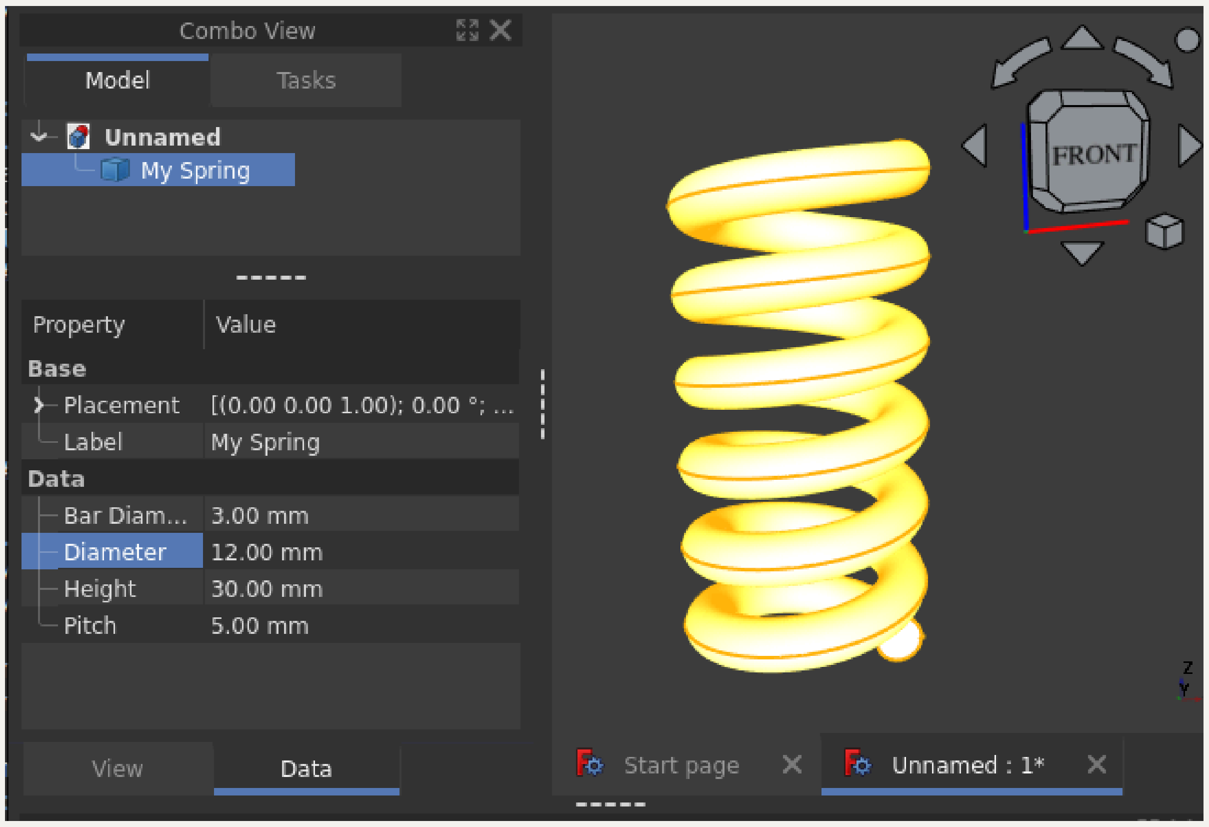
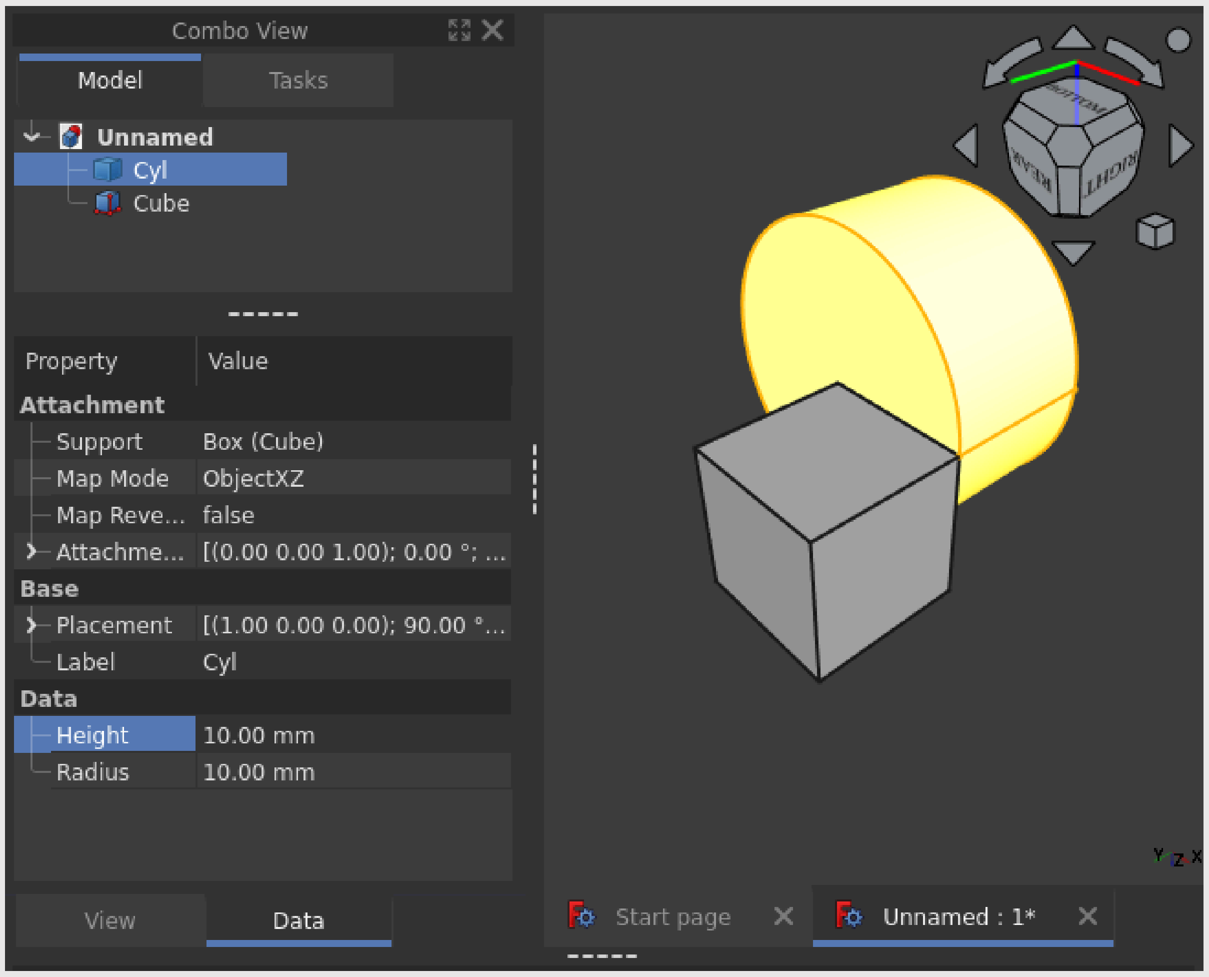
Code examples

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.

SCRIPT	DESCRIPTION	IMAGE
ex1_basic.py	example with various properties	
ex2_cube.py	part example with one Shape	

Property	Value
Base	
Label	Params
Data	
Angle	30.00 °
Elements	3
Source	
Thickness	1.00 mm

Property	Value
Base	
Placement	[(0.00 0.00 1.00); 0...
Label	Cube
Data	
Height	10.00 mm
Length	10.00 mm
Width	10.00 mm

SCRIPT	DESCRIPTION	IMAGE
ex3_spring.py	part example with one Shape	
ex4_attachable.py	part example with one attachable Shape	

Combo View
Model Tasks
Unnamed
My Spring

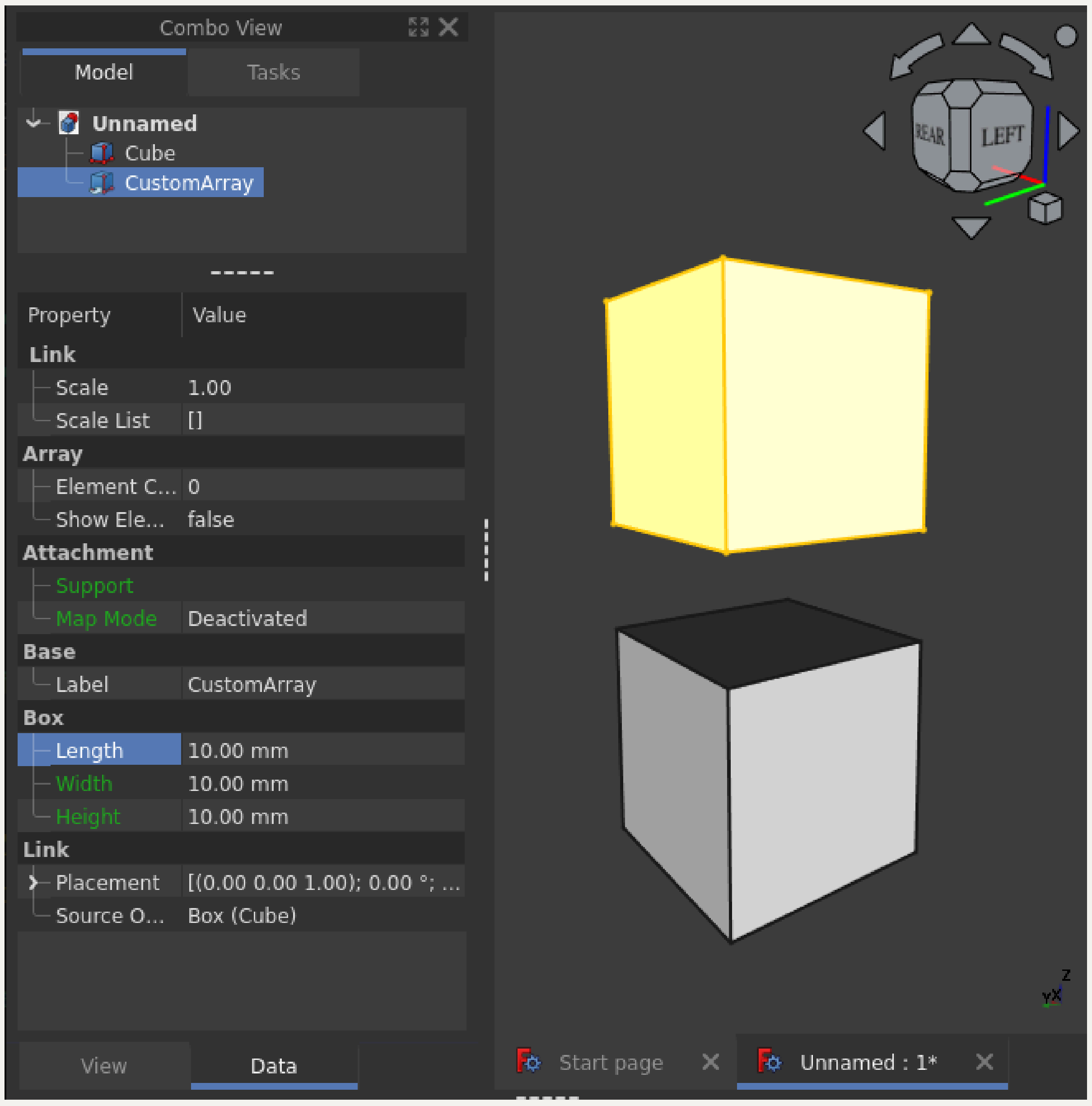
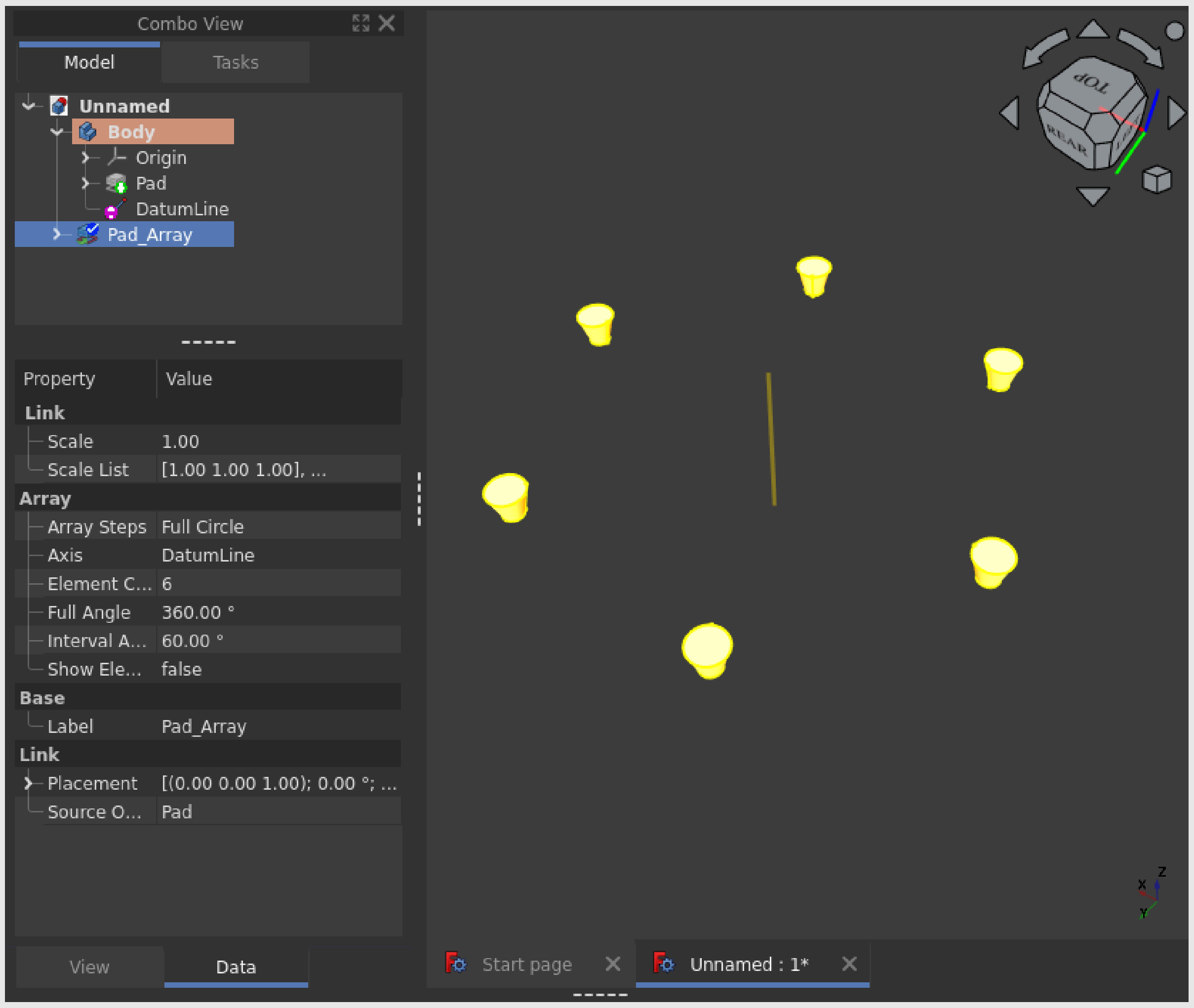
Property	Value
Base	
Placement	[(0.00 0.00 1.00); 0.00 °; ...
Label	My Spring
Data	
Bar Diam...	3.00 mm
Diameter	12.00 mm
Height	30.00 mm
Pitch	5.00 mm

View Data
Start page Unnamed : 1*

Combo View
Model Tasks
Unnamed
Cyl
Cube

Property	Value
Attachment	
Support	Box (Cube)
Map Mode	ObjectXZ
Map Reve...	false
Attachme...	[(0.00 0.00 1.00); 0.00 °; ...
Base	
Placement	[(1.00 0.00 0.00); 90.00 °...
Label	Cyl
Data	
Height	10.00 mm
Radius	10.00 mm

View Data
Start page Unnamed : 1*

SCRIPT	DESCRIPTION	IMAGE
ex5_link.py	Object with Link behavior	
ex6_link_array.py	Object with Link array behavior	

Combo View

Model | Tasks

- Unnamed
  - Cube
  - CustomArray

Property	Value
Link	
Scale	1.00
Scale List	[]
Array	
Element C…	0
Show Ele…	false
Attachment	
Support	
Map Mode	Deactivated
Base	
Label	CustomArray
Box	
Length	10.00 mm
Width	10.00 mm
Height	10.00 mm
Link	
Placement	[(0.00 0.00 1.00); 0.00 °; …
Source O…	Box (Cube)

View | Data

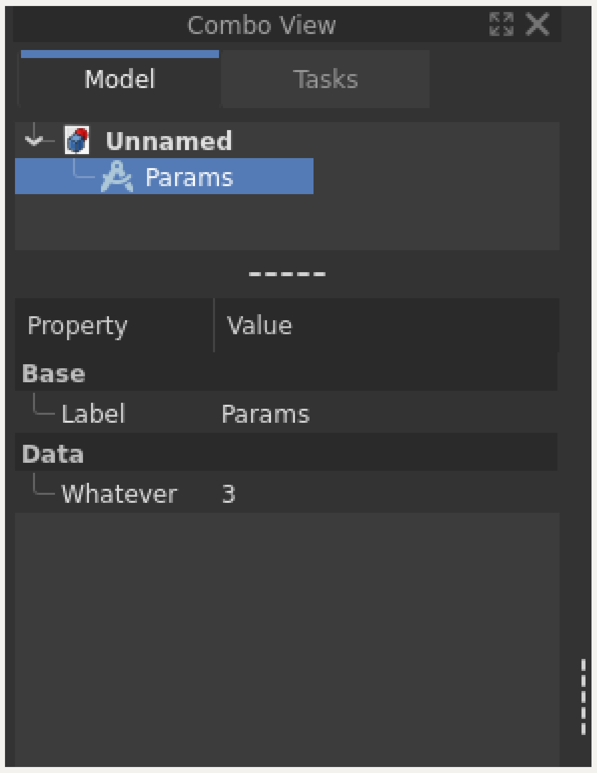
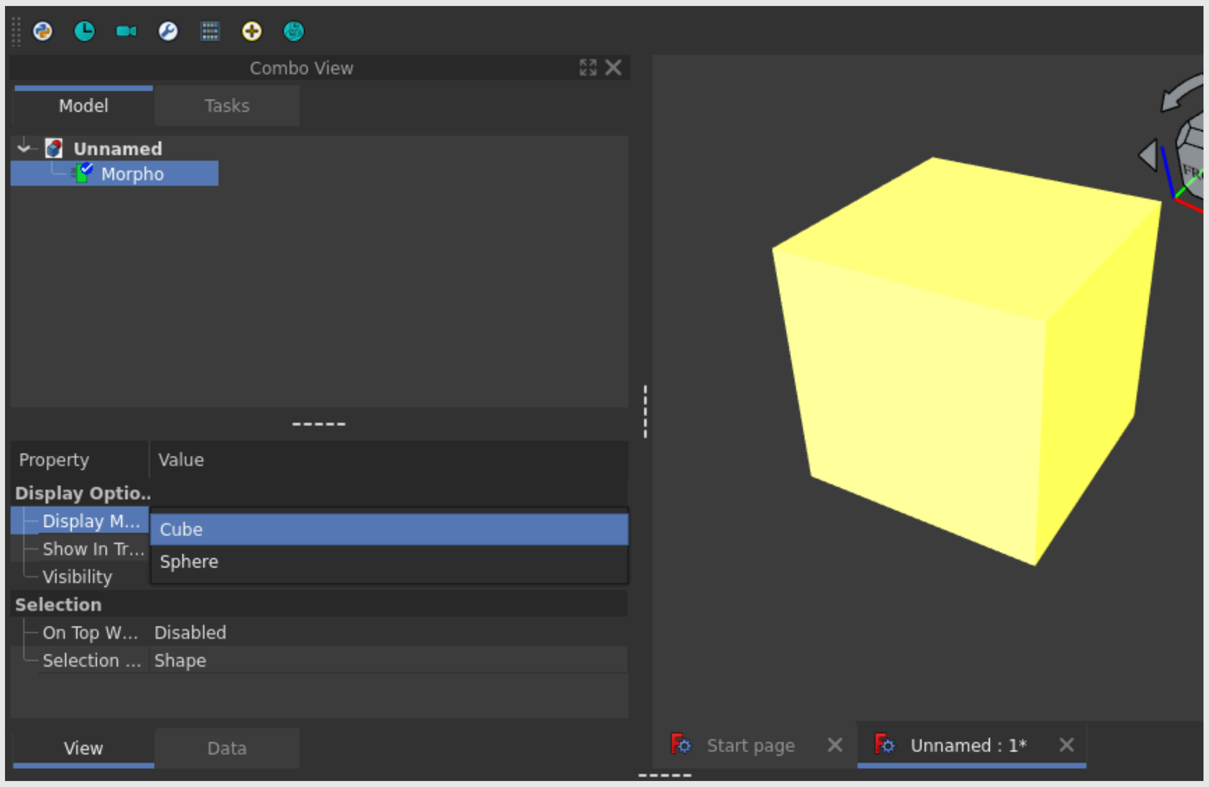
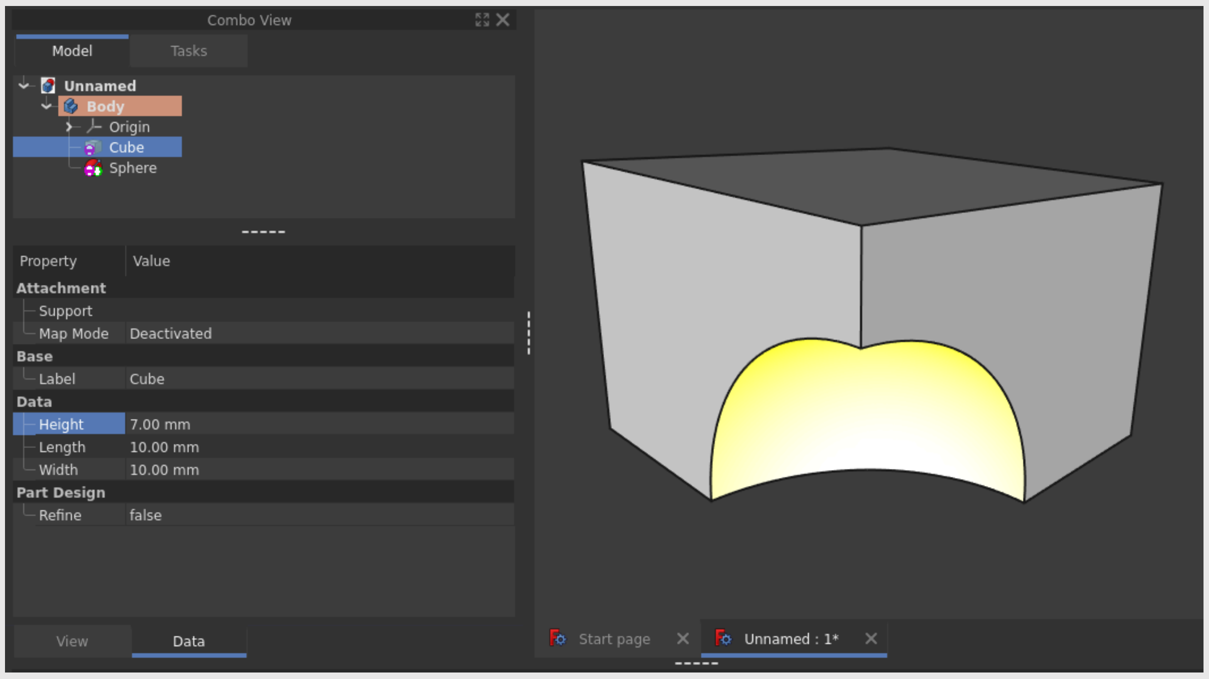
Combo View

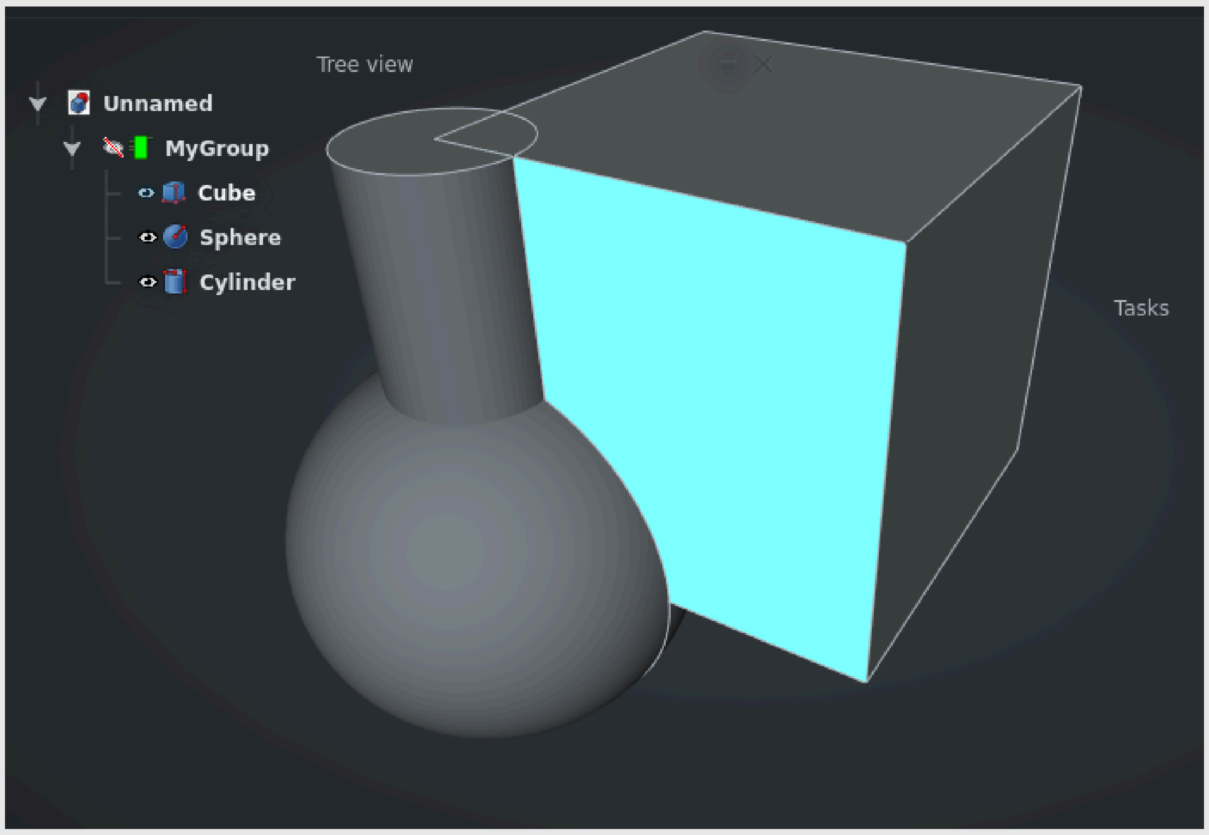
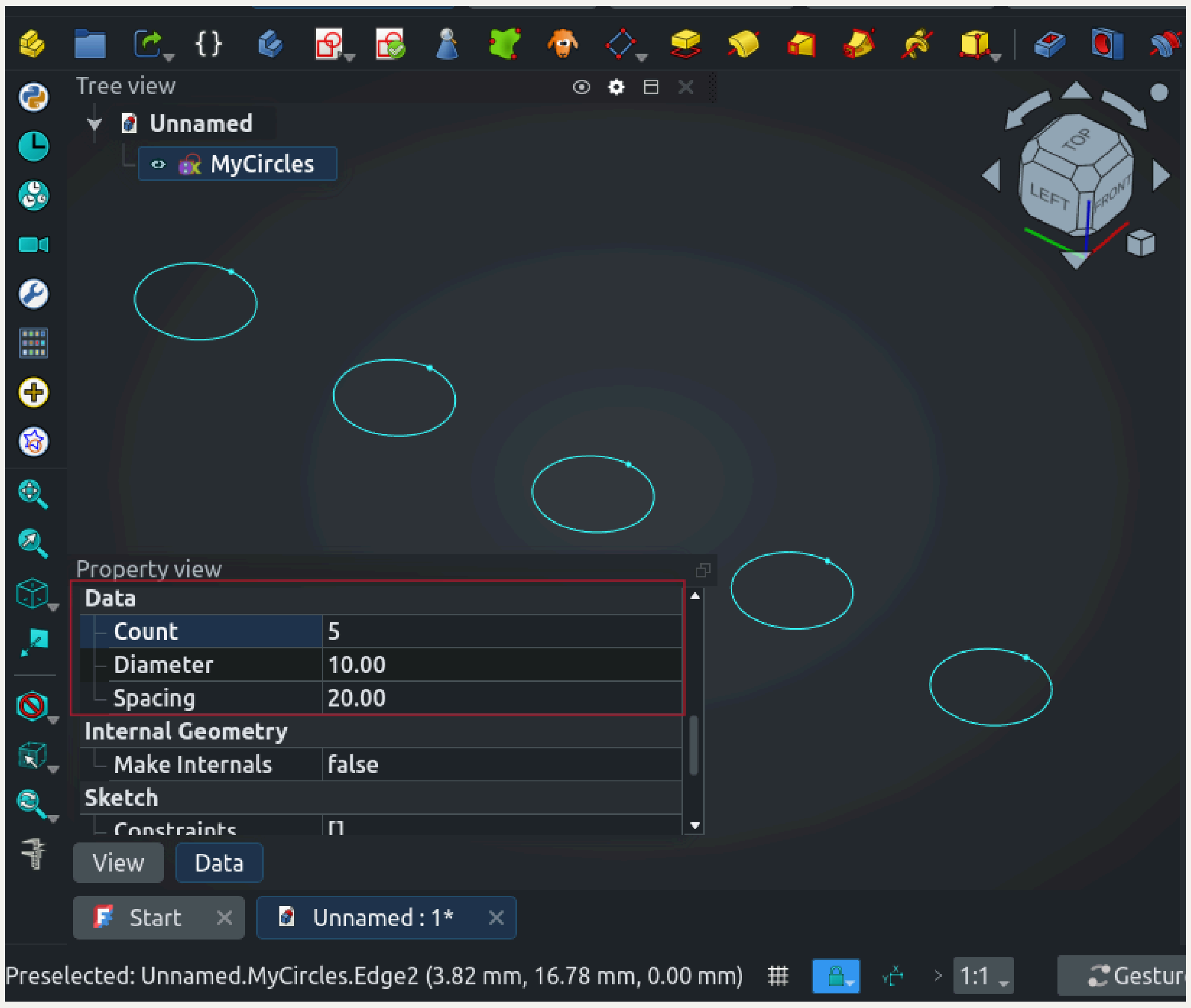
Model | Tasks

- Unnamed
  - Body
    - Origin
    - Pad
    - DatumLine
  - Pad_Array

Property	Value
Link	
Scale	1.00
Scale List	[1.00 1.00 1.00], …
Array	
Array Steps	Full Circle
Axis	DatumLine
Element C…	6
Full Angle	360.00 °
Interval A…	60.00 °
Show Ele…	false
Base	
Label	Pad_Array
Link	
Placement	[(0.00 0.00 1.00); 0.00 °; …
Source O…	Pad

View | Data

SCRIPT	DESCRIPTION	IMAGE
ex7_icon.py	Using a ViewProxy to setup the icon	 Combo View Model Tasks Unnamed Params Property Value Base Label Params Data Whatever 3
ex8_display_modes.py	Using a ViewProxy to setup display modes	 Combo View Model Tasks Unnamed Morpho Property Value Display Optio.. Display M... Cube Show In Tr... Sphere Visibility Selection On Top W... Disabled Selection ... Shape View Data Start page Unnamed : 1*
ex9_migrations.py	Basic migration	
ex10_part_design.py	Object compatible with PartDesign	 Combo View Model Tasks Unnamed Body Origin Cube Sphere Property Value Attachment Support Map Mode Deactivated Base Label Cube Data Height 7.00 mm Length 10.00 mm Width 10.00 mm Part Design Refine false View Data Start page Unnamed : 1*

SCRIPT	DESCRIPTION	IMAGE
ex11_undo_redo.py	Transactional undo/redo aware code	
ex13_docgroup.py	Group behavior	
ex14_sketch.py	Custom sketch object	

Property	Value
Data	
Count	5
Diameter	10.00
Spacing	20.00
Internal Geometry	
Make Internals	false
Sketch	
Constraints	[]

Preselected: Unnamed.MyCircles.Edge2 (3.82 mm, 16.78 mm, 0.00 mm)

Quick setup

The only required file to use this API is `fpo.py`, the key point is where to put it 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 module of the `freecad` package.

```
.
└── FreeCAD Config Dir/
    └── Mod/
        └── YourWorkbench/
            └── freecad/
                └── your_module/
                    ├── __init__.py
                    ├── init_gui.py
                    ├── fpo.py
                    ├── your_amazing_thing.py
                    └── ...
```

Usage from Macros

It is better to not define your proxy 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 proxy 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. This will make `fpo` and your modules visible and importable from FreeCAD.

Examples setup

Copy `fpo.py` and `examples/*` (the files, not the directory) into FreeCAD's *Macro* dir. then you can run the examples from the FreeCAD's Python console:

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

import ex10_part_design as ex10
ex10.create_cube_pd()

...
```

> **Important**
>
> Copy `fpo.py` in **only one place** to avoid name conflicts.