addFC - additional tools for FreeCAD

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17.07.2024

1 Goals and objectives

- Bill of materials - BOM.
- Batch processing of sheet metal parts.
- Creation of design documentation.
- Process automation.

The main purpose of the workbench is to simplify the work with large and «complex» assemblies, especially with assemblies containing sheet metal parts.

«Complex» I mean parametric models (assemblies) with a large number of objects and nodes in the form of links (App::Link). The main point is to reuse components.

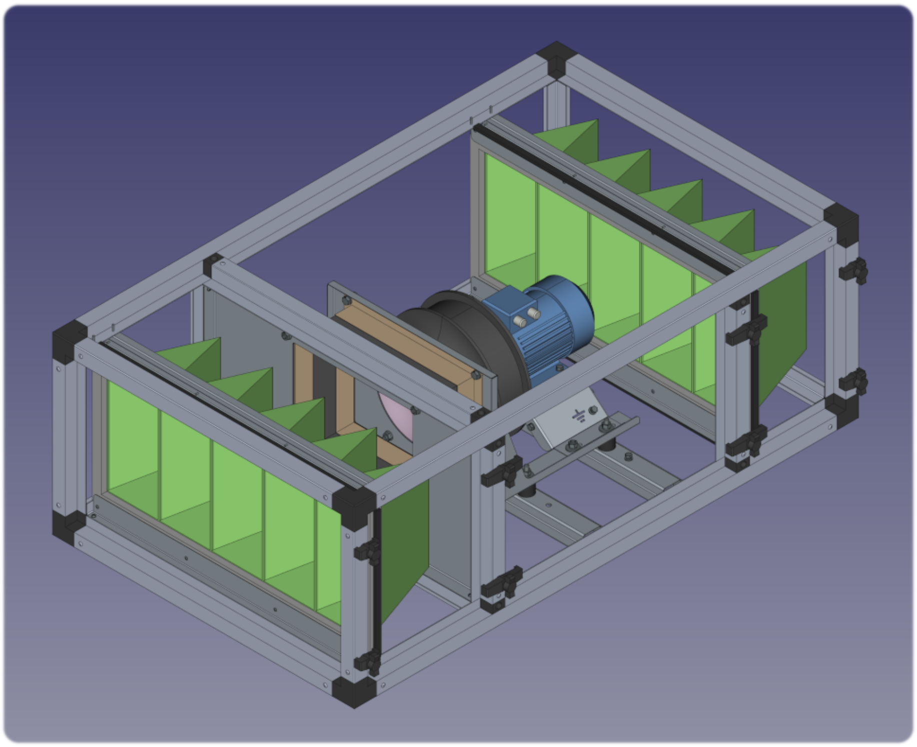


Figure 1: Example of a «complex» assembly

The logic of work is based on adding custom properties to objects, giving them certain semantic meanings.

2 Toolbar

When you select the **addFC workbench**, its toolbar will become available, it looks like this:

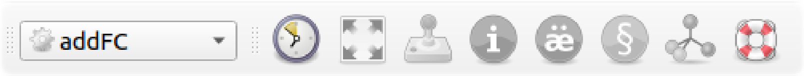


Figure 2: Toolbar

Tools in order:

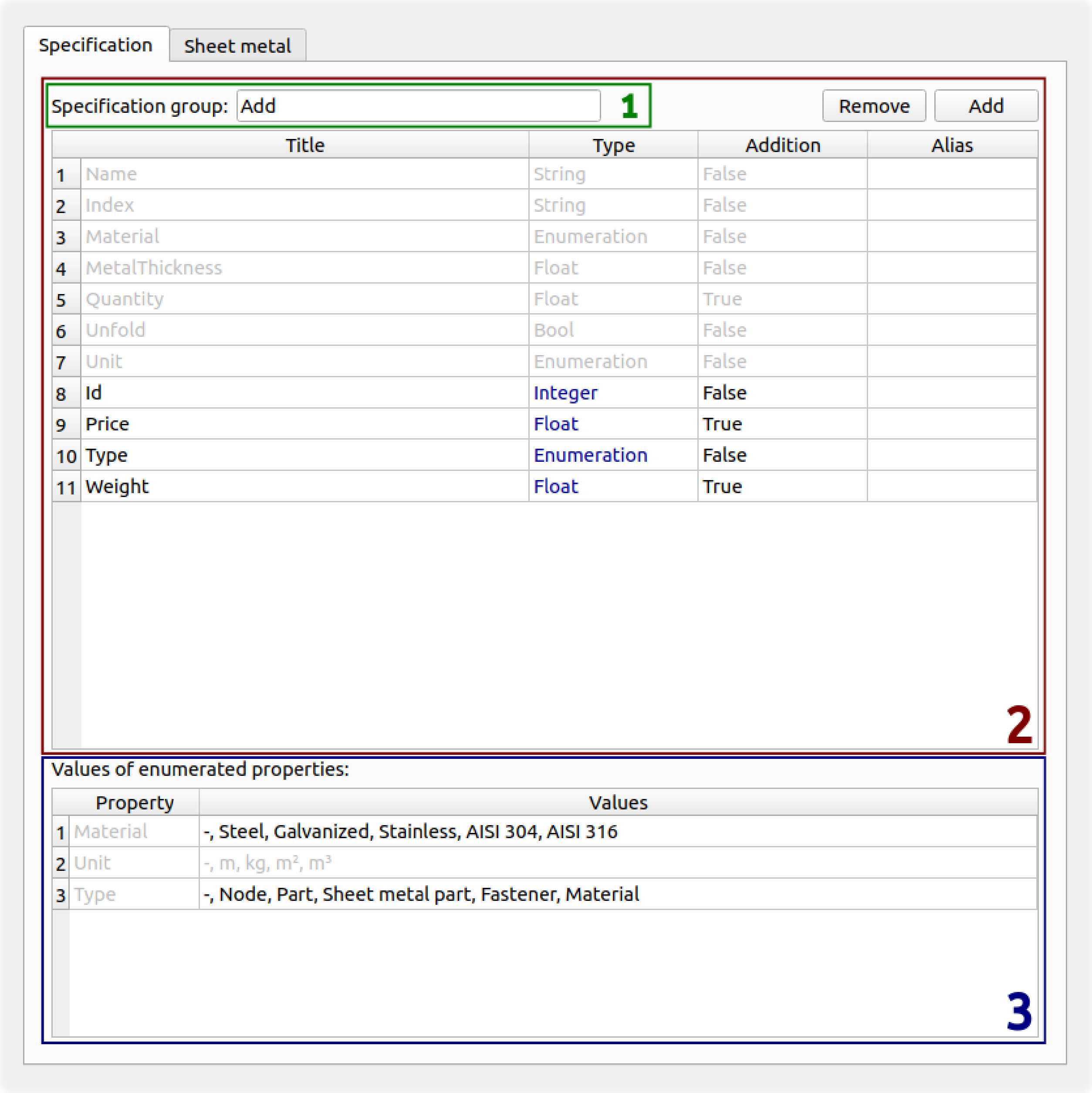
1. Open last working file - **Recent File** (Alt+Shift+R).
2. Isometry and fit all - **Display** (Alt+Shift+D).
3. Managing a parametric model - **Model Control** (Alt+Shift+C).
4. Bill of Materials (BOM) - **Specification** (Alt+Shift+S).
5. Filling an object with properties - **Add Properties** (Alt+Shift+A).
6. Creating a pipeline using coordinates - **Pipe** (Alt+Shift+P).
7. Exploded view - **Explode** (Alt+Shift+E).
8. Help - **Help and examples**.

Note: FreeCAD allows you to create additional toolbars, I recommend taking advantage of this and creating your own toolbar from the most popular functions to display it on your main workbench, for example in PartDesign.

3 Help and examples

The workbench includes samples that you can study to better understand the principles of its operation, to open one of them, use the Help and examples command on the toolbar. The most suitable example is **Assembly**, which will be discussed in this guide.

4 Preferences



Specification group: Add

	Title	Type	Addition	Alias
1	Name	String	False	
2	Index	String	False	
3	Material	Enumeration	False	
4	MetalThickness	Float	False	
5	Quantity	Float	True	
6	Unfold	Bool	False	
7	Unit	Enumeration	False	
8	Id	Integer	False	
9	Price	Float	True	
10	Type	Enumeration	False	
11	Weight	Float	True	

Values of enumerated properties:

	Property	Values
1	Material	-, Steel, Galvanized, Stainless, AISI 304, AISI 316
2	Unit	-, m, kg, m^2, m^3
3	Type	-, Node, Part, Sheet metal part, Fastener, Material

Figure 3: Specification (BOM) preferences

4.1 Area 1 - Name for grouping properties

The properties that we add to objects will be combined into a special group, the name of which can be specified in the corresponding field. This will facilitate visual perception and will not allow our properties to be mixed with standard ones.

4.2 Area 2 - User properties

This table contains all the properties available for use.

- **Title** - property name (important: Latin characters only).
- **Type** - property value type available for use:
  - **Bool** - boolean data type (true или false).
  - **Enumeration** - a list of predefined values.
  - **Float**.
  - **Integer**.
  - **String**.
- **Addition** - indicates the need to sum all property values (example: total assembly mass).
- **Alias** - a property alias, a value that will replace **Title** when displaying or exporting a specification (allows you to eliminate the restriction on Latin characters).

The **Remove** and **Add** buttons, respectively, allow you to delete a property selected in the table or add a row to create a new one.

4.3 Area 3 - Lists of predefined values

All properties with the **Enumeration** data type appear in this area.
In the **Values** column - comma-separated values to form a list.

5 Object properties

Inactive properties and values in the table are basic and required for the workbench to work correctly.

Properties should give meaning to FreeCAD objects.

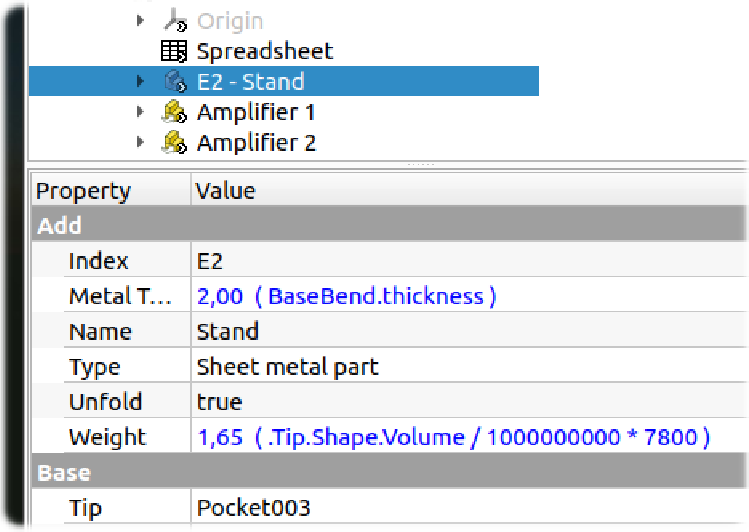
- **Name** - the name of the object is the most important property, the program works with elements only if they have a name. The name should reflect the essence of the object..
- **Index** - an identifier for determining the position of an object in an assembly and a code designation of a part for manufacturing.
- **Material** - the material of the object (enumeration). This is an important property for sheet metal, when creating a flat part view (unfold) for galvanized and stainless steel, different coefficients are used, and this property is also taken into account when saving the scan to an external file.
- **MetalThickness** - short designation: **MT**.
- **Unfold** - determines the need to create a flat view (unfold) for a specific object (relevant only for sheet metal parts).
- **Quantity** and **Unit** of measurement **(-, m, kg, m^2, m^3)**. For piece items, the default value in most cases is one, «pcs» **(-)**. Any combination is available for different materials: for example, a seal length of **1.2 m** or a quantity of insulation of **4.2 m^2**. Important: values are summed for objects of the same name.

5.1 Additional properties

These properties are not essential (they can be removed), but nevertheless they are useful in work:

- **Id** - some object identifier for communication with another program.
- **Price** - cost of the object.
- **Type** - object type (enumeration). Useful property for grouping elements when displaying or exporting a BOM.
- **Weight** - object mass.

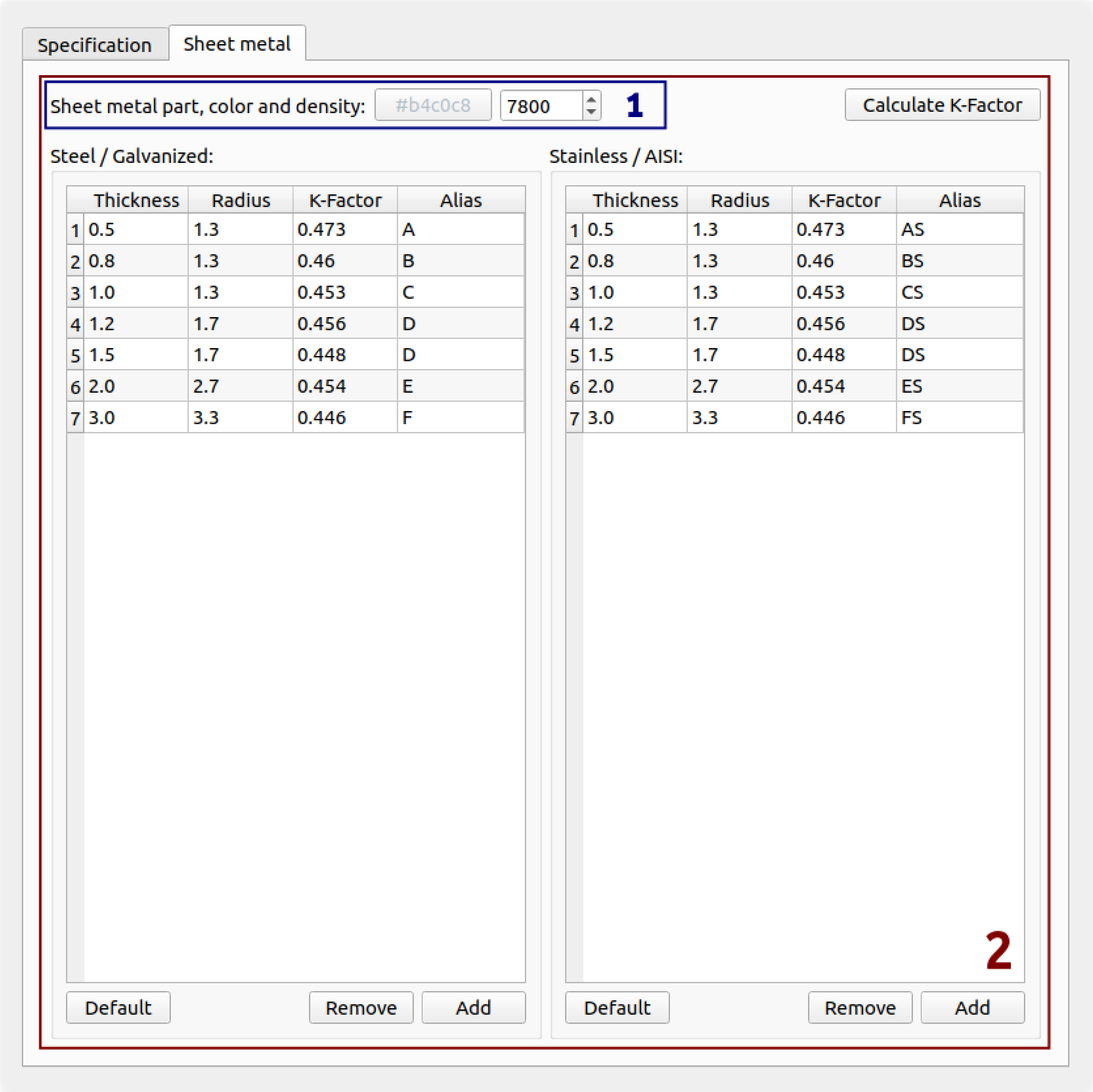
To account for an object by the program, only the ***Name*** *property is required, all others are used as needed.*



- Origin
- Spreadsheet
- E2 - Stand
- Amplifier 1
- Amplifier 2

Property	Value
Add	
Index	E2
Metal T...	2,00 (BaseBend.thickness)
Name	Stand
Type	Sheet metal part
Unfold	true
Weight	1,65 (.Tip.Shape.Volume / 1000000000 * 7800)
Base	
Tip	Pocket003

Figure 4: Example of an object with filled in properties



Sheet metal part, color and density: #b4c0c8 7800

Steel / Galvanized:

	Thickness	Radius	K-Factor	Alias
1	0.5	1.3	0.473	A
2	0.8	1.3	0.46	B
3	1.0	1.3	0.453	C
4	1.2	1.7	0.456	D
5	1.5	1.7	0.448	D
6	2.0	2.7	0.454	E
7	3.0	3.3	0.446	F

Stainless / AISI:

	Thickness	Radius	K-Factor	Alias
1	0.5	1.3	0.473	AS
2	0.8	1.3	0.46	BS
3	1.0	1.3	0.453	CS
4	1.2	1.7	0.456	DS
5	1.5	1.7	0.448	DS
6	2.0	2.7	0.454	ES
7	3.0	3.3	0.446	FS

Figure 5: Sheet metal preferences

5.2 Area 4 - Sheet metal part options

The first value is the color of the object in hex format (`#b4c0c8`), the second is the average value of steel density (7800 kg/m^3).

5.3 Area 5 - Sheet steel parameters

This table indicates the acceptable sheet metal **thickness** for use and their parameters, such as the internal bending **radius**, the **k-factor** used when calculating the flat type (unfold) and the **alias** for the metal thickness, it is necessary for correct export to external format.

The **Calculate K-Factor** button automatically calculates the **k-factor** for each thickness using formulas from the strength of materials:

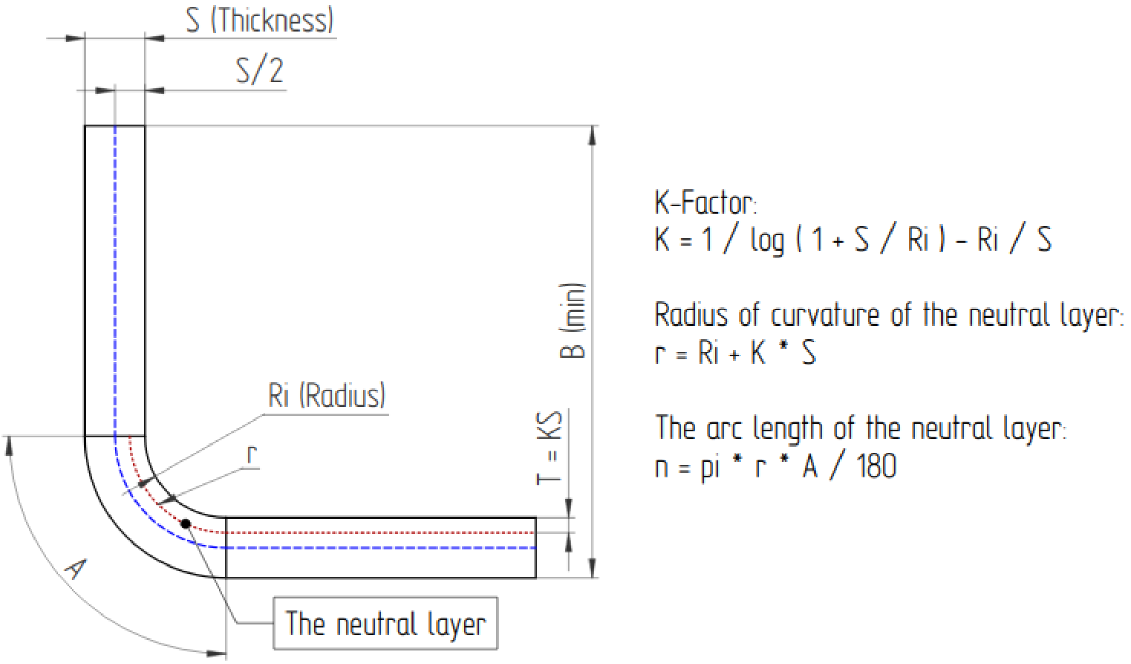
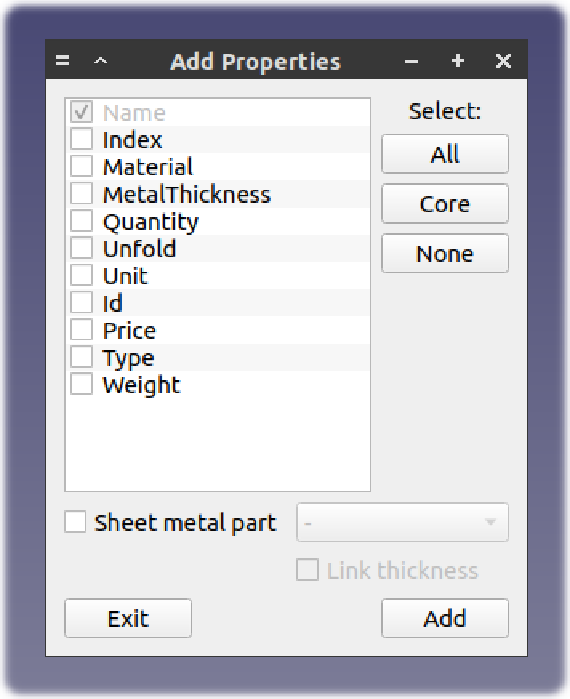

S (Thickness)
S/2
B (min)
Ri (Radius)
r
T = KS
A
The neutral layer
K-Factor:
K = 1 / log (1 + S / Ri) - Ri / S
Radius of curvature of the neutral layer:
r = Ri + K * S
The arc length of the neutral layer:
n = pi * r * A / 180

Figure 6: Formulas for calculating sheet metal parameters

6 Filling an object with properties

To add properties, you need to select one or more objects and use the Add Properties command on the toolbar.



☑ Name
□ Index
□ Material
□ MetalThickness
□ Quantity
□ Unfold
□ Unit
□ Id
□ Price
□ Type
□ Weight

□ Sheet metal part -

□ Link thickness

Figure 7: Interface **Add Properties**

The entire list of available custom properties is visible in the command interface. You need to mark the ones you need and click **Add**.

Buttons **All**, **Core**, **None** - select all properties, only the main ones and clear the selection.

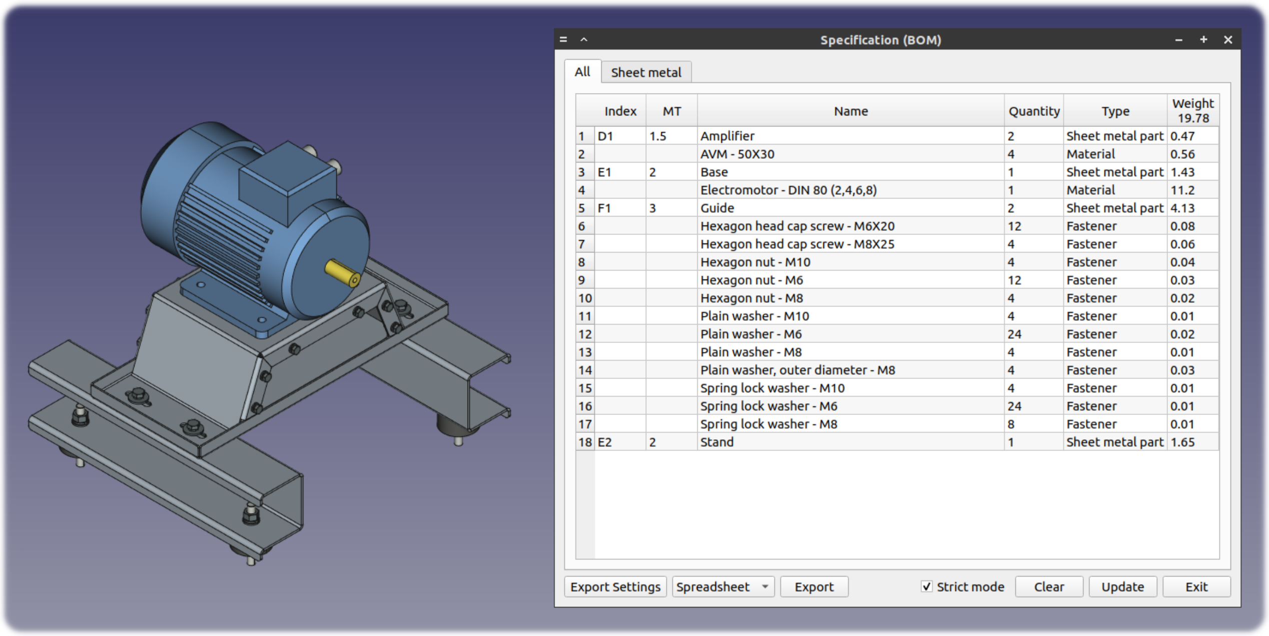
The **Sheet metal part** checkbox will check all the necessary properties for a sheet metal part, allow you to select the type of material and, if desired, associate the **MetalThickness** property with the thickness parameters of the object. Additionally, the element will be assigned a weight and color based on the parameters specified in the settings - fihure 5: area 1.

Note: During the process of assigning a **Name** and **Index**, the program tries to guess the property values based on the **Label** of the object.

To automatically fill these properties, the name template must match: **«Index. Name»** or **«Index. Name - Copy»**. If the template matches, the values will be filled in correctly - example.

7 Bill of materials - BOM

To generate and work with BOM, you must use the Specification command on the toolbar. Based on user properties, the program will generate a specification for any model (assembly), consider an example from the workbench:

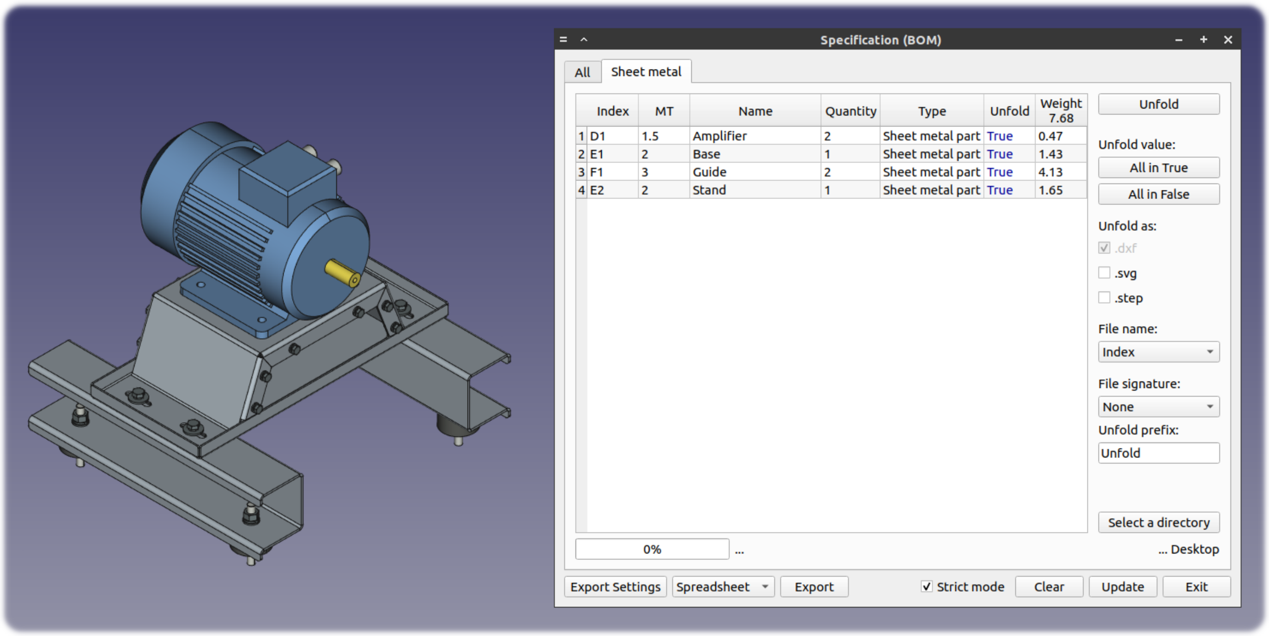


Specification (BOM)

All | Sheet metal

	Index	MT	Name	Quantity	Type	Weight 19.78
1	D1	1.5	Amplifier	2	Sheet metal part	0.47
2			AVM - 50X30	4	Material	0.56
3	E1	2	Base	1	Sheet metal part	1.43
4			Electromotor - DIN 80 (2,4,6,8)	1	Material	11.2
5	F1	3	Guide	2	Sheet metal part	4.13
6			Hexagon head cap screw - M6X20	12	Fastener	0.08
7			Hexagon head cap screw - M8X25	4	Fastener	0.06
8			Hexagon nut - M10	4	Fastener	0.04
9			Hexagon nut - M6	12	Fastener	0.03
10			Hexagon nut - M8	4	Fastener	0.02
11			Plain washer - M10	4	Fastener	0.01
12			Plain washer - M6	24	Fastener	0.02
13			Plain washer - M8	4	Fastener	0.01
14			Plain washer, outer diameter - M8	4	Fastener	0.03
15			Spring lock washer - M10	4	Fastener	0.01
16			Spring lock washer - M6	24	Fastener	0.01
17			Spring lock washer - M8	8	Fastener	0.01
18	E2	2	Stand	1	Sheet metal part	1.65

Figure 8: Bill of materials



Specification (BOM)

All | Sheet metal

	Index	MT	Name	Quantity	Type	Unfold	Weight 7.68
1	D1	1.5	Amplifier	2	Sheet metal part	True	0.47
2	E1	2	Base	1	Sheet metal part	True	1.43
3	F1	3	Guide	2	Sheet metal part	True	4.13
4	E2	2	Stand	1	Sheet metal part	True	1.65

Unfold value:

Unfold as: ☑ .dxf □ .svg □ .step

File name: Index

File signature: None

Unfold prefix: Unfold

0% ... ... Desktop

Figure 9: Bill of materials, sheet metal parts

The specification interface contains two tabs:

- **All** - all objects.
- **Sheet metal** - sheet metal objects.

The **Strict mode** option - if the checkbox is unchecked, the program will process all user properties in your group - figure 3: area 1, not only those specified in the table (area 2).

I think everything is clear with the **All** tab, let's look at **Sheet metal** objects - this tab contains functions for their batch processing. The manufacturing process for such parts will in most cases require two elements:

- **Blank (unfold)** - a flat view of an object for nesting and processing on machines.
- **Part in 3D format (step)** - for bending sheet metal.

All parts from the list generated based on the model (assembly), depending on the value of the **Unfold** property, can be processed and exported to external files, such as dxf, svg (unfolding) and step (3D).

Select a directory - to save work results (the default value is the user's desktop).

Unfold prefix - the name of the directory in which the files will be saved, as well as an option for signing the part.

File signature - list of options for part signature. The signature is text in the file, inside the outline of the part, which can be useful when nesting. At the moment the function is only available for dxf format. When set to **None**, the signature is disabled.

File Name - the template by which the files will be named, for example, for the part **«E2 - Stand»** (figure 4) the names will be as follows:

- **Index** = E2 (1).dxf
- **Name** = Stand (1).dxf
- **Index + Name** = (E2) Stand (1).dxf

The number in brackets at the end of the name is the copy number, if there are two or more identical parts in the assembly, they will be saved in separate files.

After selecting the desired options and parameters, you can click the **Unfold** button and the program will save all received data in the specified path. The work process can be observed in the progress indicator and in FreeCAD, in the report panel (report view).

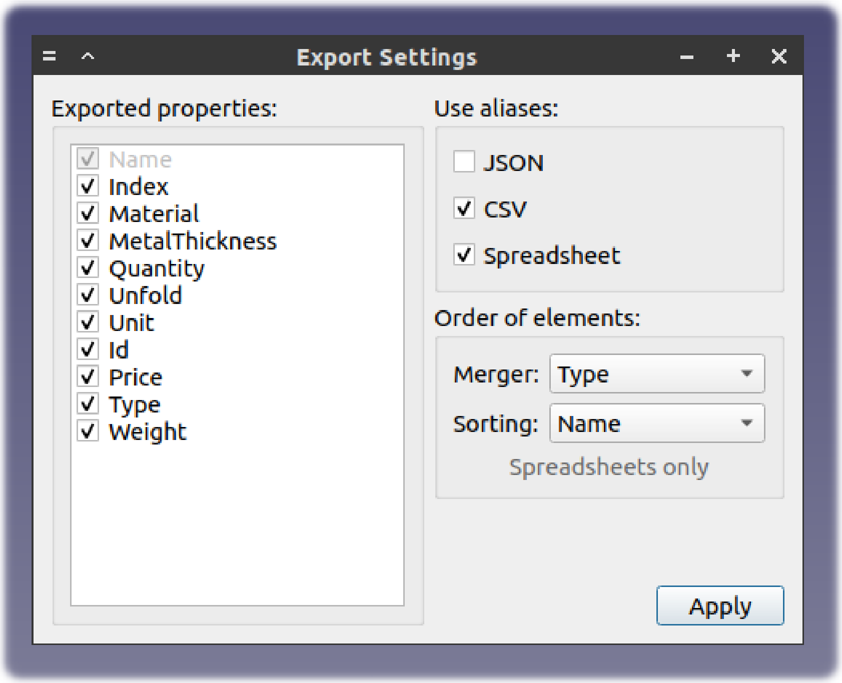
Important: the part files will be placed in additional directories, the names of which correspond to the **Alias** of the steel thickness - figure 5: area 2.

8 Export specification

The program can export a model (assembly) specification for subsequent viewing, editing or other use, available formats:

- **Spreadsheet** - FreeCAD spreadsheet workbench.
- **json** - the most suitable option for subsequent automation.
- **csv** - comma-separated values.

First, let's look at the export options, button **Export Settings**.



Exported properties:

☑ Name
☑ Index
☑ Material
☑ MetalThickness
☑ Quantity
☑ Unfold
☑ Unit
☑ Id
☑ Price
☑ Type
☑ Weight

Use aliases:

□ JSON
☑ CSV
☑ Spreadsheet

Order of elements:

Merger: Type
Sorting: Name
Spreadsheets only

Figure 10: Specification export options

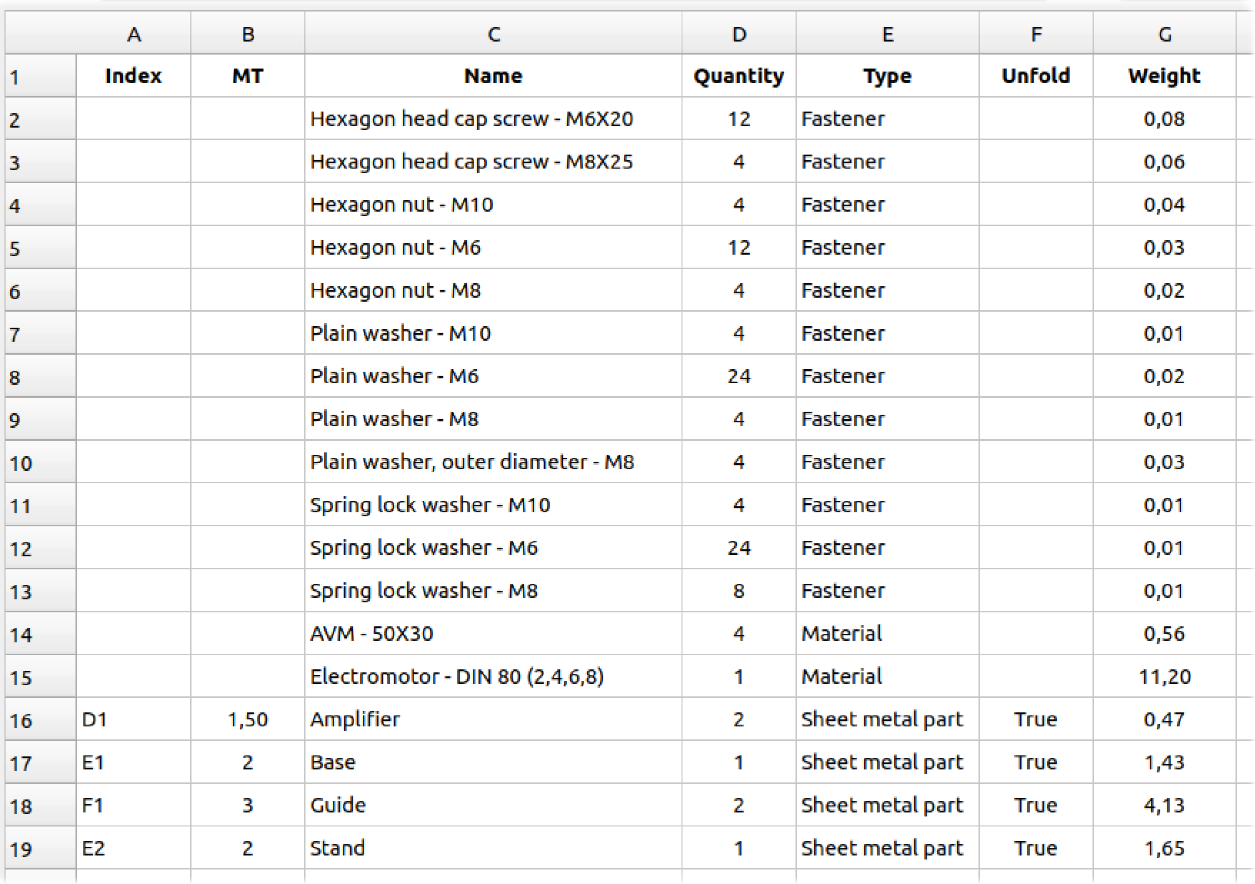
On the left side of the interface you need to select the properties that will be available for export.

In the **Use aliases** area, you need to mark the formats in which aliases will be used, as a replacement for the **Title** property.

In the **Order of elements** area, you need to specify properties for grouping and sorting objects:

- **Merger** - a property by the value of which the elements will be grouped, the most suitable is the object **Type**, for example, display first all fasteners, then materials, then sheet metal parts.
- **Sorting** - a property by the value of which objects will be sorted within a **Merger**, the most logical thing is to sort by **Index** or **Name**.

Select the necessary options, the appropriate format and click the button **Export**.



	A	B	C	D	E	F	G
1	**Index**	**MT**	**Name**	**Quantity**	**Type**	**Unfold**	**Weight**
2			Hexagon head cap screw - M6X20	12	Fastener		0,08
3			Hexagon head cap screw - M8X25	4	Fastener		0,06
4			Hexagon nut - M10	4	Fastener		0,04
5			Hexagon nut - M6	12	Fastener		0,03
6			Hexagon nut - M8	4	Fastener		0,02
7			Plain washer - M10	4	Fastener		0,01
8			Plain washer - M6	24	Fastener		0,02
9			Plain washer - M8	4	Fastener		0,01
10			Plain washer, outer diameter - M8	4	Fastener		0,03
11			Spring lock washer - M10	4	Fastener		0,01
12			Spring lock washer - M6	24	Fastener		0,01
13			Spring lock washer - M8	8	Fastener		0,01
14			AVM - 50X30	4	Material		0,56
15			Electromotor - DIN 80 (2,4,6,8)	1	Material		11,20
16	D1	1,50	Amplifier	2	Sheet metal part	True	0,47
17	E1	2	Base	1	Sheet metal part	True	1,43
18	F1	3	Guide	2	Sheet metal part	True	4,13
19	E2	2	Stand	1	Sheet metal part	True	1,65

Figure 11: BOM export result

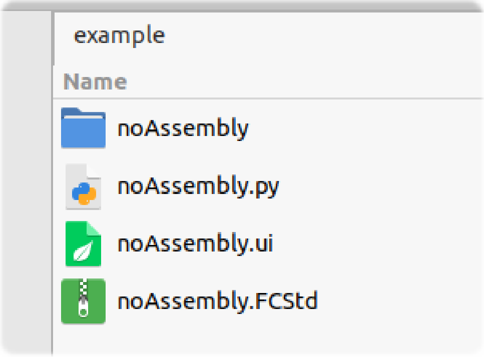
9 Managing a parametric model

The Model Control command is available on the taskbar, the purpose of which is to launch a control program for a **parametric** model.

In my work, I was convinced that not one of the existing (for FreeCAD) assembly systems in combination with tables and equations is capable of providing such capabilities that are available from program code.

I write control files and interfaces for my parametric models, which are convenient to call with one command. To do this, next to the main model (assembly) file there should be two files named similarly to the main one.

In the samples supplied with the workbench, a simple example of a parametric model is available for study - `addFC/repo/example`



example

Name

noAssembly

noAssembly.py

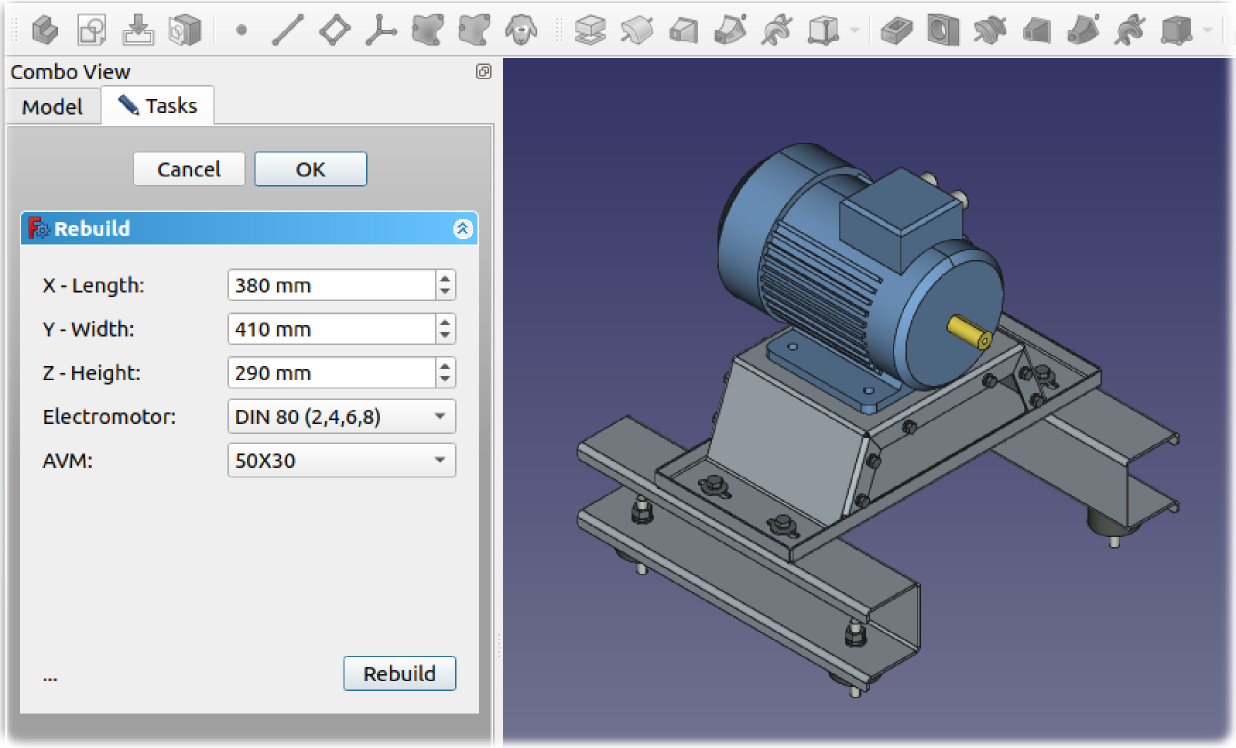
noAssembly.ui

noAssembly.FCStd

Figure 12: Parametric model files

- **.FCStd** - main model file - assembly.
- **.ui** - user interface - Qt.
- **.py** - source code - Python.
- **noAssembly** - directory with additional files.

Having opened the main file, you can call its control program using the Model Control command:



X - Length: 380 mm

Y - Width: 410 mm

Z - Height: 290 mm

Electromotor: DIN 80 (2,4,6,8)

AVM: 50X30

Figure 13: Parametric model management interface

For convenience, the graphical user interface is built into the FreeCAD sidebar; after setting the necessary parameters and selecting components from the lists, click the **Rebuild** button - the model will be updated.

10 Creating a pipeline using coordinates

To be continued ...

11 Exploded view

To be continued ...