

## **FreeCAD Basics (Vaasa Hacklab 17.10.2017)**

**Installation:** <https://www.freecadweb.org/wiki/Download>

### **Starting the program and a new file:**

- Blank Window -> View (Top Bar) -> Workbench -> Part Design
- File -> New or Create new document icon (below the file )

### **User interface:**

- Combo View (Left Left Screen)
  - The model tab is a model tree
  - The content of the Tasks is what is being done. Tasks for displaying the current tool or suggestions for tools if nothing is just being used.
- 3D View
  - Displays the model graphically
  - You can select and rotate / pan / zoom with the mouse
    - Recommended to change navigation settings: right mouse button in 3D view -> Navigation Styles -> Blender
    - Select items: Left mouse button (Ctrl)
    - Rotation: Center mouse button (roll)
    - Panning: Shift + Center Mouse (Roll)
    - Zoom: Rotate mouse wheel OR Ctrl + Shift + right mouse button and move mouse up / down
  - Toggle axis cross -> Toggle axis cross
- The Toolbar
  - Common File Features (New, Open, Save ...)
  - Workbench drop-down menu
  - (A macro)
  - View Tools
  - Workbench-specific tools

### **Sketching:**

- The construction of a model is usually started by creating a new sketch feature
- Create sketch (Tasks tab or toolbar)
  - The program asks the level for which the feature is to be created. One of the basic levels (XY, XZ or YZ levels) or offset can be selected.
  - The OK -> program switches to the sketch view
- Creating drawing elements
  - Create line -> start drawing, left mouse button -> stop drawing, left mouse button
  - The tool remains active, so you can continue drawing. Where is the end point turns green. Draw the line the same way as the first one.
  - You can exit the tool by pressing the right mouse button or the Esc key
  - Multiple consecutive lines available with the Polyline tool
  - The other drawing elements are added in the same way
- constraints
  - It is (yeah) completely unambiguous
  - The program automatically creates constraints if Auto constraints is on. After drawing, the

- shape is thus limited, but also in certain directions. (left mouse button down on target)
- Limitations are divided into two categories, geometric and measurement constraints. You should limit the format as far as possible, making editing easier later.
- To add restrictions, first select the item (s) you want to restrict. The new Restriction icon appears if the items are checked correctly. Otherwise, the program will provide an error message and instructions on what to choose.
- Dimensions are added accordingly. Often, there are several options for sizing, and the way they do, for example, is the Triangle.
- Whenever the need is, they are the only ones to be able to do it. The Tasks tab shows the number of degrees if there are or are "Fully constrained sketch" if the shape is fully restricted.
- On the other hand, if too many constraints are the odds, and the number of the faulty constraint / constraint. These should be corrected before leaving the sketch mode.
- The exit tab on the Tasks tab

### Creating 3D features:

- When a sketch feature is created, it can be used for example
  - Select the desired sketch feature from the Tasks tab or toolbar.
- An extruded song appears in the 3D view
  - Usually, the extrusion is given a measure
  - The extrusion can also be done in two different directions in a different amount
  - Symmetric to plane splits the extrusion evenly on both sides of the sketch level
- When the extrusion is desired, the feature is the OK on the Tasks tab. After that, the model tree shows a pad feature below which you can find a sketch of the previous one.
  - In this case, there is a dependency between the features.
- You can add a piece of flat surface
  - Move the cursor over the surface surface, and select it with the left mouse button
  - Tasks tab or toolbar -> Create sketch
    - You can link to the linking tool
    - Select the linking tool from the toolbar and show the lines you want to link
    - The lines appear in the form of violets and cannot be edited by them
  - When the new sketch feature is finished, select it again.
  - The new 3D feature will appear in the sketch will be found underneath it
- Borders (Chamfer, Blend)
  - Select the desired edge / edges of the chamfer function
  - The border features do not require a sketch as a base
  - The Tasks tab is given by pressing OK
  - The new 3D feature will appear in the template tree
  - Blend works the same way
- Removing material from a song is done with the Pocket function
  - The function is able to make pockets and holes in the piece
  - Works like Pad
  - Through the all-type function, the song always makes a hole in the song, even if the song thickness is changed later. (The hole can be made by the depth of the piece.)
- Mirror feature, Linear pattern ...
  - You can also add to the paragraph by mirroring or copying an existing feature
  - Mirror Function or the Mirror Function
    - The program mirrors the feature of a sketch-level horizontal or vertical axis of the underlying feature or a user-defined line
  - The Linear pattern works in a similar way

- Specifications must specify the number of copies (user-defined line)
- Polar pattern is the feature of the line
- More complex copying (eg grid) can be done with Multitransform
- Revolve-feature
  - 3D features can also be done by rotating the symmetrical tracks
  - Works the same way as extruding, ie selecting the Revolve function
    - The shape can be spun around the horizontal or vertical axis of the sketch feature
    - The default is 360 °, but it can also be done with a smaller degree
- The primitives
  - General basic forms (cube, ball, cone ...) can be found in Part-Workbench
  - These features do not require a sketch as a base, but are defined by marine parameters. As a result, they still loose .

### **Boolean operations:**

- Boolean operations need to be done more easily with extruding and spinning, or if, for example, extrusion should be done on a curved surface
- The whole model can be done using the Boolean Operations, but it is often too obscure and difficult to use. In addition, the features are not dependent on other features.
- Adding a Boolean Mission
  - The model must have the two separate pieces to make the boolean
  - Operations can be found in Part-Workbench
  - The toolbar or the Boolean tool is the toolbar or the Boolean tool.
  - With the operation, the songs can be joined together, the song of the song.
- After that, the model tree shows a new feature under the operation .

### **Exporting a song in .stl format for 3D printing:**

- Select the desired song from the template tree (the song is activated in 3D view) -> File -> Export -> Files type -> STL Mesh
- Stl is also a mesh file, so the song is exported with limited accuracy. Edit -> Preferences -> Import-Export -> Mesh Formats
- Usually, the export works well, but there are no errors. In this case, you can try to fix the song with, for example, Blender or some automatic Fixer .

