

# Tvorba 3D modelů - FreeCAD

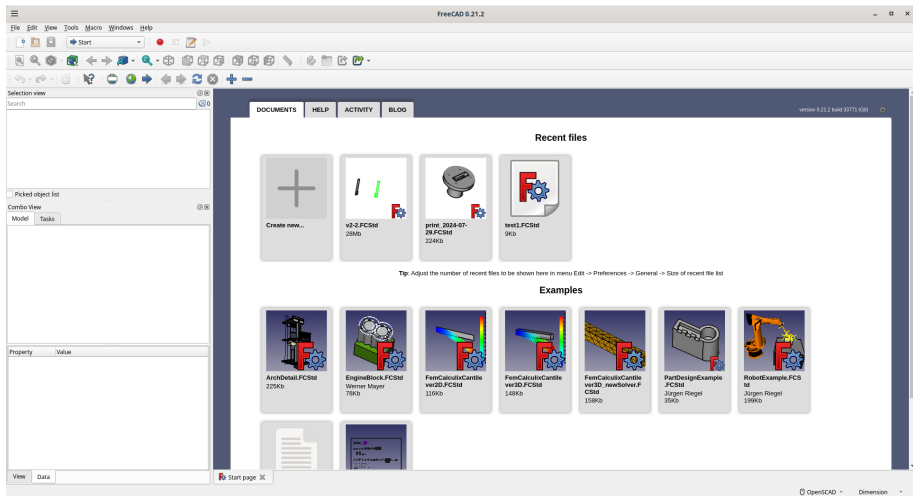
- <https://www.freecad.org>
- různé operační systémy
- zdrojový kód
- manuály a návody
- příklady
- videa s návody

# Ovládání

- menu a mouse
- různá pracovní prostředí - workbench
- řada přídavných prostředí
- Python

# FreeCAD využití

- 3D modely
- technické výkresy 2D, 3D
- robotika
- stavebnictví
- simulace FEM - metoda konečných prvků
- ...



# Help

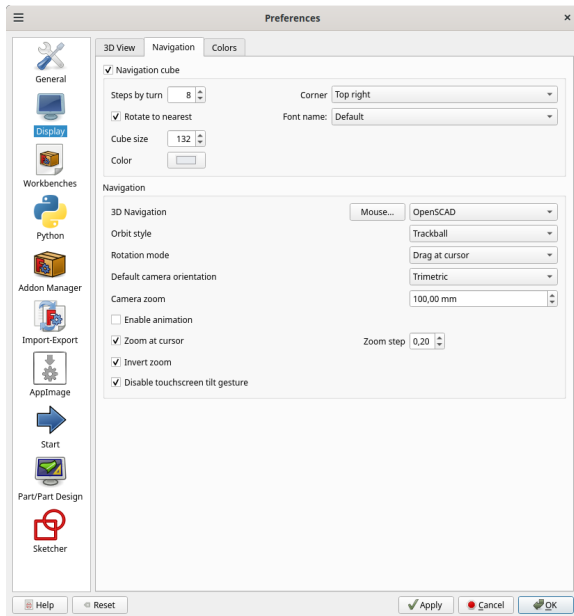
The screenshot shows the FreeCAD 0.21.2 interface with the Help window open. The window title is 'FreeCAD 0.21.2'. The menu bar includes 'File', 'Edit', 'View', 'Tools', 'Macro', 'Part', 'Measure', 'Windows', and 'Help'. The toolbar contains various icons for file operations, navigation, and modeling. The left sidebar has a 'Selection view' search box, a 'Picked object list', a 'Combo View' with 'Model' and 'Tasks' tabs, and a 'Property' table. The main content area displays the 'Sketcher geometries' help page.

## Sketcher geometries

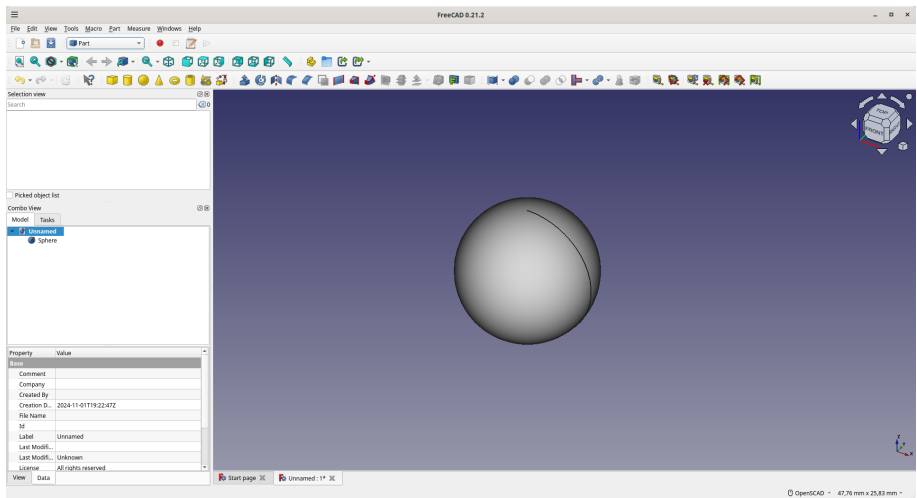
These are tools for creating objects.

- **Point:** Creates a point.
- **Polyline:** Creates a series of line and arc segments connected by their endpoints. The tool has several modes.
- **Line:** Creates a line, [introduced in version 1.0](#): The tool has three modes.
- **Create arc:**
  - **Arc by center:** Creates an arc by its center and its endpoints, [introduced in version 1.0](#): Or by its endpoints and a point along the arc.
  - **Arc by 3 points:** Creates an arc by its endpoints and a point along the arc, [introduced in version 1.0](#): This is the same tool as **Arc by center** but with a different initial mode.
  - **Arc of ellipse:** Creates an arc of ellipse.
  - **Arc of hyperbola:** Creates an arc of hyperbola.
  - **Arc of parabola:** Creates an arc of parabola.
- **Create circle/ellipse:**
  - **Circle by center:** Creates a circle by its center and a point along the circle, [introduced in version 1.0](#): Or by three points along the circle.

# Nastavení



# Workbench - Part



# Workbench - Part Design - Sketcher

The screenshot displays the FreeCAD 0.21.2 interface in the Sketcher workbench. The main workspace shows a dark blue background with a green rectangular sketch. Red dimension lines indicate a width of 60 mm and a height of 30 mm. The sketch is fully constrained, as indicated by the status bar and the 'Fully constrained' message in the solver messages panel.

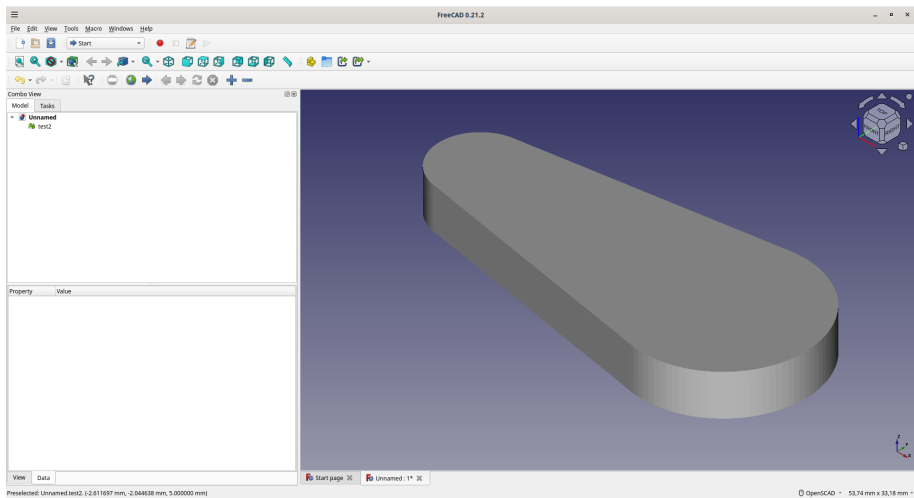
The interface includes a menu bar (File, Edit, View, Tools, Macro, Sketch, Windows, Help), a toolbar with various sketching tools, and several panels on the left:

- Selection view:** Search field and object list.
- Picked object list:** Empty.
- Combo View:** Model and Tasks tabs.
- Solver messages:** Shows 'Fully constrained'.
- Constraints:** Lists seven constraints (Constraint1 to Constraint7), all checked.
- Elements:** Lists three elements (1-Line, 2-Line, 3-Line), all checked.

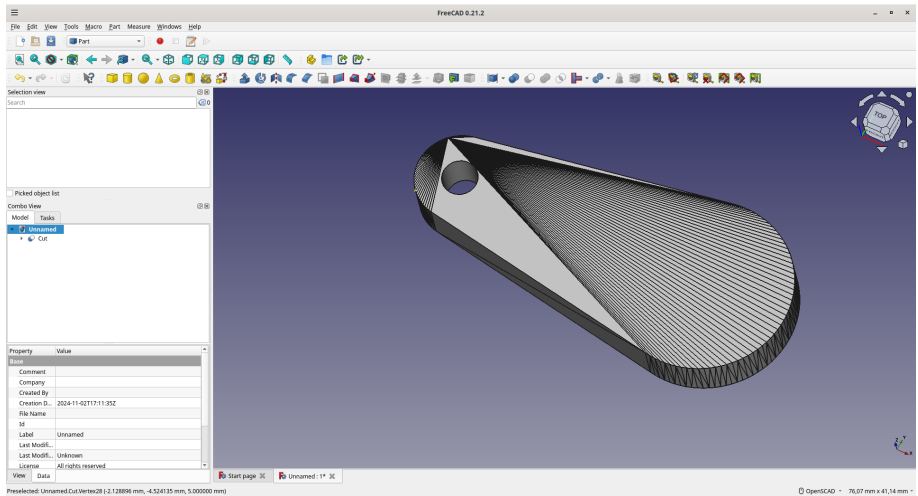
The status bar at the bottom shows 'Preselected: Unnamed.Body.Sketch.Edge4 (0.000000 mm,11.152897 mm,8.000000 µm)' and 'OpenSCAD - 101,48 mm x 54,88 mm'.



# Import STL



# Import STL - Create shape - Convert to solid



# Workbench - OpenSCAD

FreeCAD 0.21.2

File Edit View Tools Macro OpenSCAD Windows Help

OpenSCAD

Selection view

Search

Picked object list

Combo View

Model Tasks

Close

Add OpenSCAD Element

Add Refresh Open... Save... Clear c

as Mesh

```
$f=100;
hull()
  translate([30,0,0]) cylinder(d=20, h=5);
  cylinder(d=10, h=5);

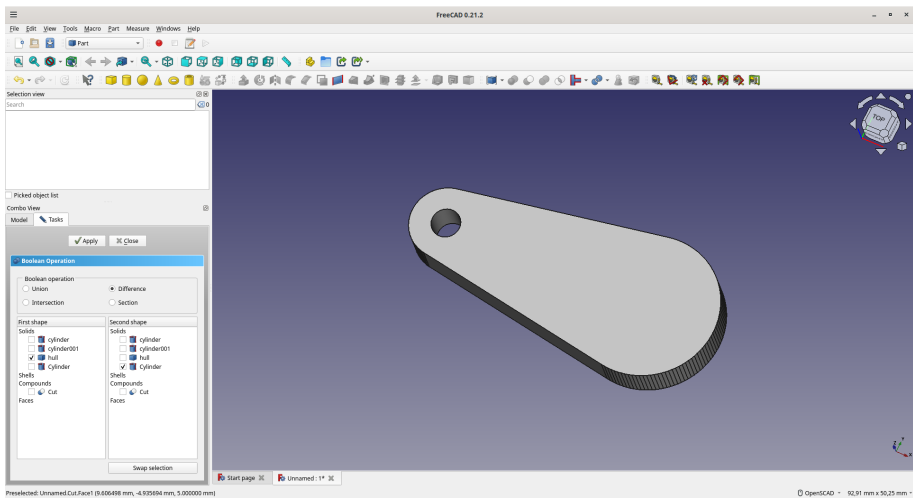
```

Start page Unnamed: 1\*

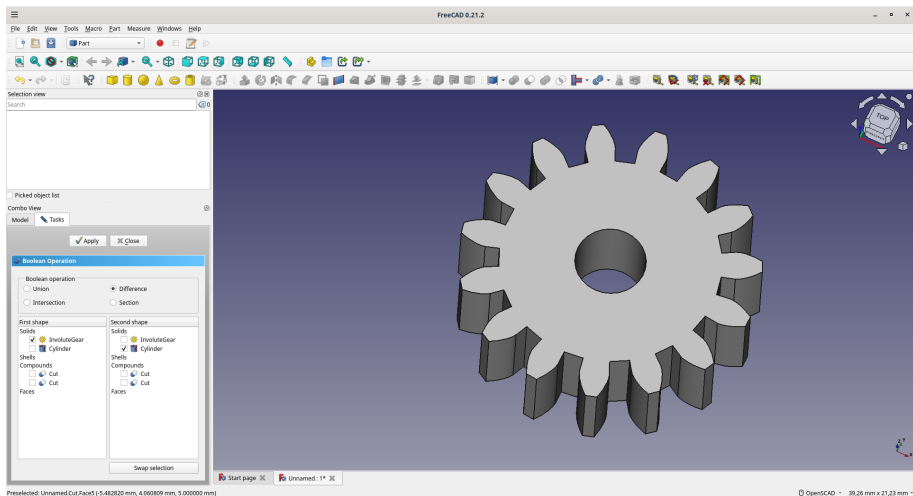
Preselected: Unnamed.hull.Face#0 (0.535326 mm, -5.161280 mm, 2.911218 mm)

OpenSCAD - 91,53 mm x 49,50 mm

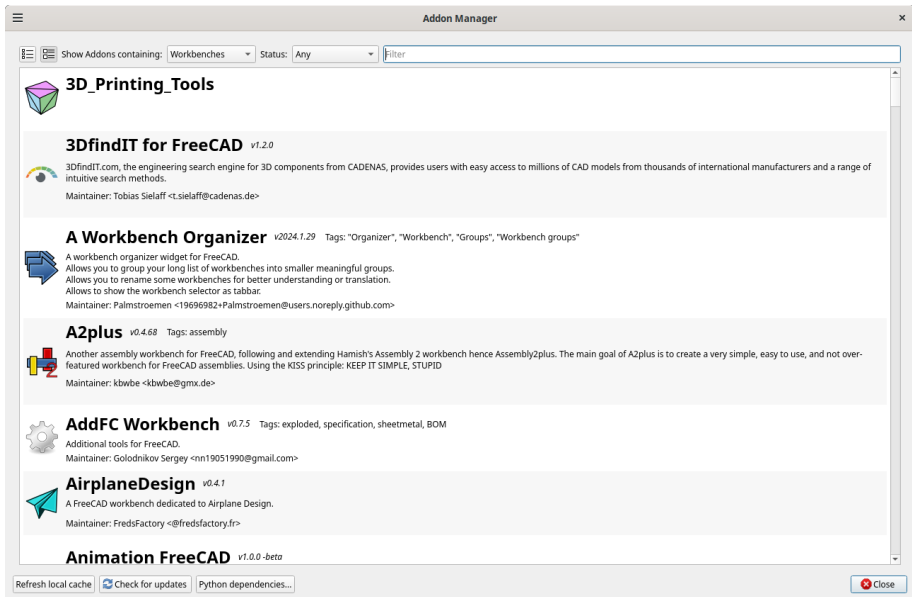
# Workbench - OpenSCAD



# Workbench - Gear



# Workbench - přidání



The screenshot shows the 'Addon Manager' window in FreeCAD. At the top, there are search filters: 'Show Addons containing: Workbenches', 'Status: Any', and a 'Filter' text box. Below the filters, a list of workbenches is displayed, each with an icon, a title, version, tags, description, and maintainer information.

- 3D\_Printing\_Tools**
- 3DfindIT for FreeCAD** v1.2.0  
3DfindIT.com, the engineering search engine for 3D components from CADENAS, provides users with easy access to millions of CAD models from thousands of international manufacturers and a range of intuitive search methods.  
Maintainer: Tobias Sielaff <t.sielaff@cadenas.de>
- A Workbench Organizer** v2024.1.29 Tags: "Organizer", "Workbench", "Groups", "Workbench groups"  
A workbench organizer widget for FreeCAD.  
Allows you to group your long list of workbenches into smaller meaningful groups.  
Allows you to rename some workbenches for better understanding or translation.  
Allows to show the workbench selector as tabbar.  
Maintainer: Palmstroemen <19696982+Palmstroemen@users.noreply.github.com>
- A2plus** v0.4.68 Tags: assembly  
Another assembly workbench for FreeCAD, following and extending Hamish's Assembly 2 workbench hence Assembly2plus. The main goal of A2plus is to create a very simple, easy to use, and not over-featured workbench for FreeCAD assemblies. Using the KISS principle: KEEP IT SIMPLE, STUPID  
Maintainer: kbwbe <kbwbe@gmx.de>
- AddFC Workbench** v0.7.5 Tags: exploded, specification, sheetmetal, BOM  
Additional tools for FreeCAD.  
Maintainer: Golodnikov Sergey <nn19051990@gmail.com>
- AirplaneDesign** v0.4.1  
A FreeCAD workbench dedicated to Airplane Design.  
Maintainer: FredsFactory <@fredsfactory.fr>
- Animation FreeCAD** v1.0.0 -beta

At the bottom of the window, there are buttons for 'Refresh local cache', 'Check for updates', 'Python dependencies...', and 'Close'.

# Zaoblení hran

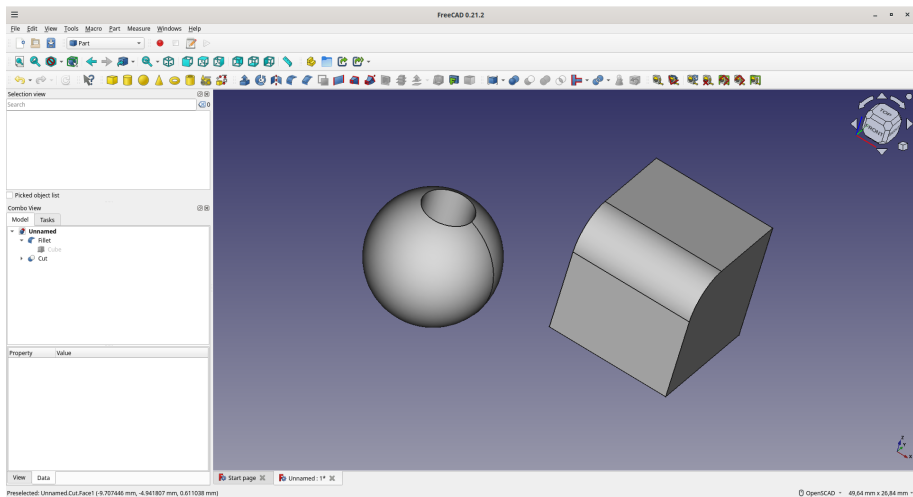
The screenshot displays the FreeCAD 0.21.2 interface. The main 3D view shows a grey cube with rounded top edges. The left sidebar contains several panels:

- Selection view:** Shows the selected object as "Unnamed#Box.Edge10 (Cube)".
- Picked object list:** Currently empty.
- Combo View:** Shows a tree view with "Unnamed" expanded to "Fillet", and "Cube" selected.
- Property table:** Lists properties for the selected object.

Property	Value
<b>Attachment</b>	
Support	
Map Mode	Deactivated
<b>Base</b>	
Placement	[(0,0,0,0,1,00); 0,00 °; (0,00 mm, 0,00 mm, 0,00 mm)]
<b>Box</b>	
Label	Cube
Length	10,00 mm
Width	10,00 mm
Height	10,00 mm

At the bottom, the status bar shows "Preselected: Unnamed Fillet.Face2 (2.513644 mm, 0.000000 mm, 2.694568 mm)" and "OpenSCAD - 39,12 mm x 21,16 mm".

# Více objektů





# Draft - Array

The screenshot displays the FreeCAD 0.21.2 software interface. The main 3D view shows a grid of 20 gray cylinders arranged in a 4x5 pattern. The interface includes a menu bar (File, Edit, View, Tools, Macro, Drafting, Annotation, Modification, Utilities, Windows, Help), a toolbar with various icons, and a left-hand sidebar with several panels: Selection view, Picked object list, Combo View (showing 'Unnamed' and 'Array'), and Property Value (showing metadata like Comment, Company, Created By, File Name, Label, License, etc.). The status bar at the bottom indicates the current selection: 'Preselected: UnnamedArrayFace1 (0.809988 mm, -1.817612 mm, 4.211879 mm)'. The bottom right corner shows system tray icons and window management controls.

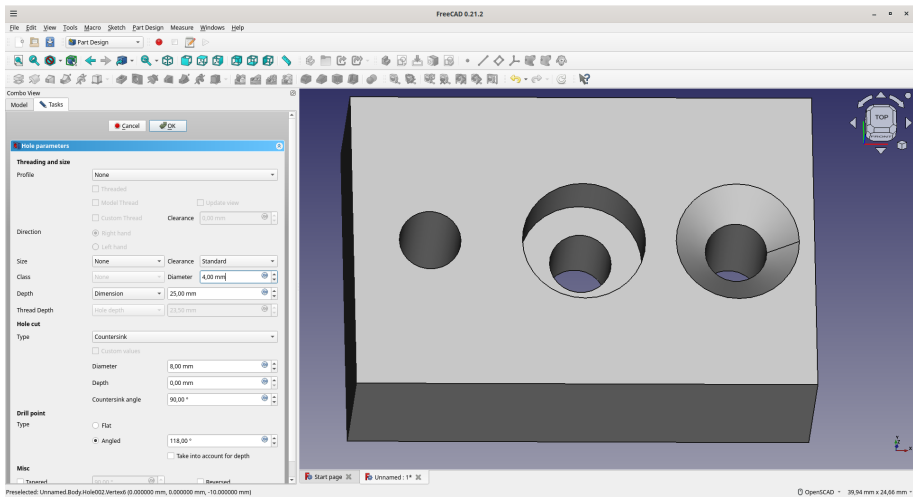
# Zrcadlení

The screenshot displays the FreeCAD 0.21.2 interface. The main 3D view shows a mirrored cone model. The top half of the cone is colored green, and the bottom half is grey. The software interface includes a menu bar (File, Edit, View, Tools, Macro, Part, Measure, Windows, Help), a toolbar with various CAD tools, and a Combo View tree on the left. The Property panel on the left shows the following data:

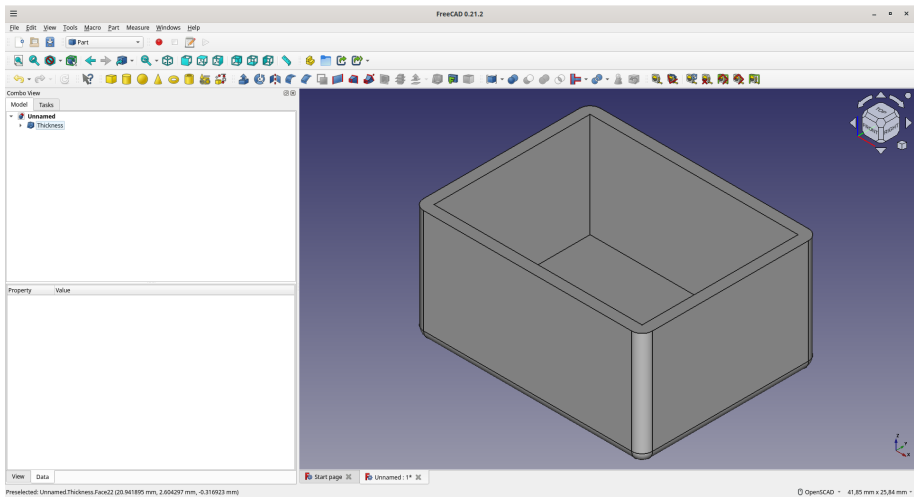
Property	Value
<b>Attachment</b>	
Support	
Map Mode	Deactivated
<b>Base</b>	
Placement	[0,00,0,00,1,00]; 0,00 °; (0,00 mm, 0,00 mm, 0,00 mm)
Label	Cone
<b>Cone</b>	
Radius1	0,00 mm
Radius2	4,00 mm
Height	10,00 mm
Angle	300,00 °

The status bar at the bottom indicates: Preselected: Unnamed Cone.Face1 [-0.343267 mm, -1.044895 mm, 2.756671 mm] and OpenSCAD - 44,56 mm x 27,51 mm.

# Otvory



# Thickness



- **Příklady:**

`https://wiki.freecad.org/Tutorials`

- **Videa s návody:**

`https://www.youtube.com/results?search\_query=freecad+3d+example`