



FreeCAD

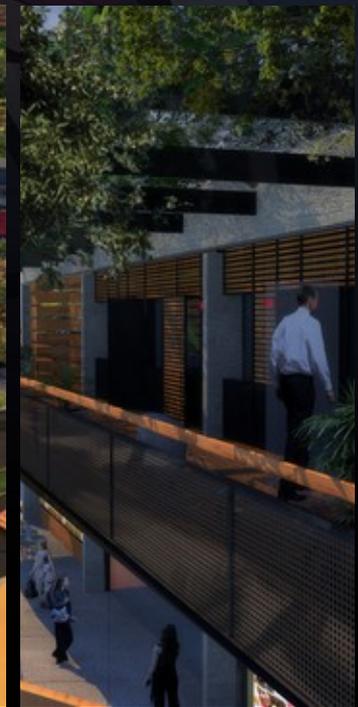
Uma plataforma de design
feita para construir

Yorik van Havre
SEMCITEC GUARULHOS 2015

- Por favor tuite com #FreeCAD
- Interrompa quando quiser
- Perdoe os erros de português
(e as piadas prontas)
- PDF desta palestra em <http://yorik.uncreated.net>

Yorik van Havre

- Arquiteto (de casas, não de informação!), e um dos desenvolvedores do FreeCAD
- <http://yorik.uncreated.net>
- yorik@uncreated.net
- [@yorikvanhavre](#)



<http://www.uncreated.net>

O que é o FreeCAD

- Aplicativo de modelagem 3D
- Software livre (licença LGPL):
Gratuito, aberto, comunidade
muito dinâmica
 - Multiplataforma (Mac,
Windows, Linux)

- Feito para modelar “qualquer coisa para ser construído no mundo real”
- Muito foco na precisão, dimensões reais, padrões
- Paramétrico (a forma dos objetos é controlada por parâmetros, como “Altura” ou “Largura”). Isso permite modificações fáceis depois.

- Multiparadigma (serve para muitos tipos de uso: impressão 3D, engenharia, arquitetura,...)
- Desenvolvido por uma comunidade de programadores, usuários e outros apaixonados
- Python: Fácil de estender, modificar, escrever scripts, etc.

Combo View [Icons]

Model Tasks

Labels & Attributes

Application

Property Value

View Data



Your version of FreeCAD is up to date.

Start a new project

- Getting started
- Part Design**
- Architectural Design
- Ship Design
- Work with Meshes

Recent Files

Latest news

Downloading latest news...

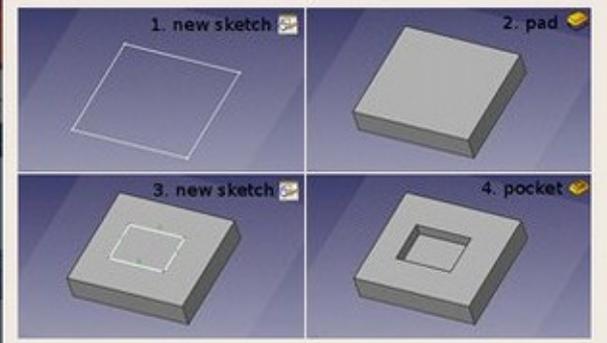
On the web

- FreeCAD Homepage
- User manual
- Tutorials
- Python resources
- The FreeCAD-tutorial blog

Designing parts

The Part Design workbench is designed to create complex pieces based on constrained 2D sketches. Use it to draw 2D shapes, constrain some of their elements and extrude them to form 3D pieces.

Example workflow :



Start page [Close]

Python console [Icons]

```
Python 2.7.10 (default, Sep 13 2015, 20:30:50)
[GCC 5.2.1 20150911] on linux2
Type 'help', 'copyright', 'credits' or 'license' for more information.
>>> import WebGui
>>> from StartPage import StartPage
>>> WebGui.openBrowserHTML(StartPage.handle(), 'file:/// + App.getResourceDir() + 'Mod/Start/StartPage/', 'Start page')
>>>
```



Combo View

Model Tasks

Labels & Attributes

Application

Property Value

View Data

Python console

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>>>
```

- Arch
- Assembly
- Complete
- Draft
- Drawing
- FEM
- Image
- Inspection
- Mesh Design
- <none>
- OpenSCAD
- Part Design
- Part
- Path
- Plot
- Points
- Raytracing
- Reverse Engineering
- Robot
- Ship
- Sketcher
- Spreadsheet
- Start**
- Test framework
- Web

Start a new project

- Getting started
- Part Design**
- Architectural Design
- Ship Design
- Work with Meshes

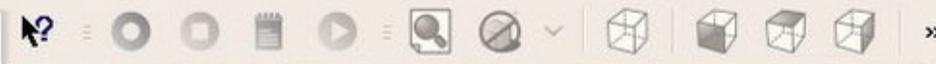
Recent Files

Latest news

Downloading latest news...

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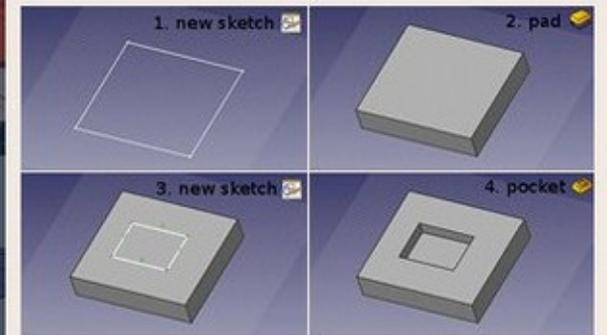
Your version of FreeCAD is up to date.

Start Center

Designing parts

The **Part Design** workbench is designed to create complex pieces based on constrained 2D sketches. Use it to draw 2D shapes, constrain some of their elements and extrude them to form 3D pieces.

Example workflow :



Start page



Combo View

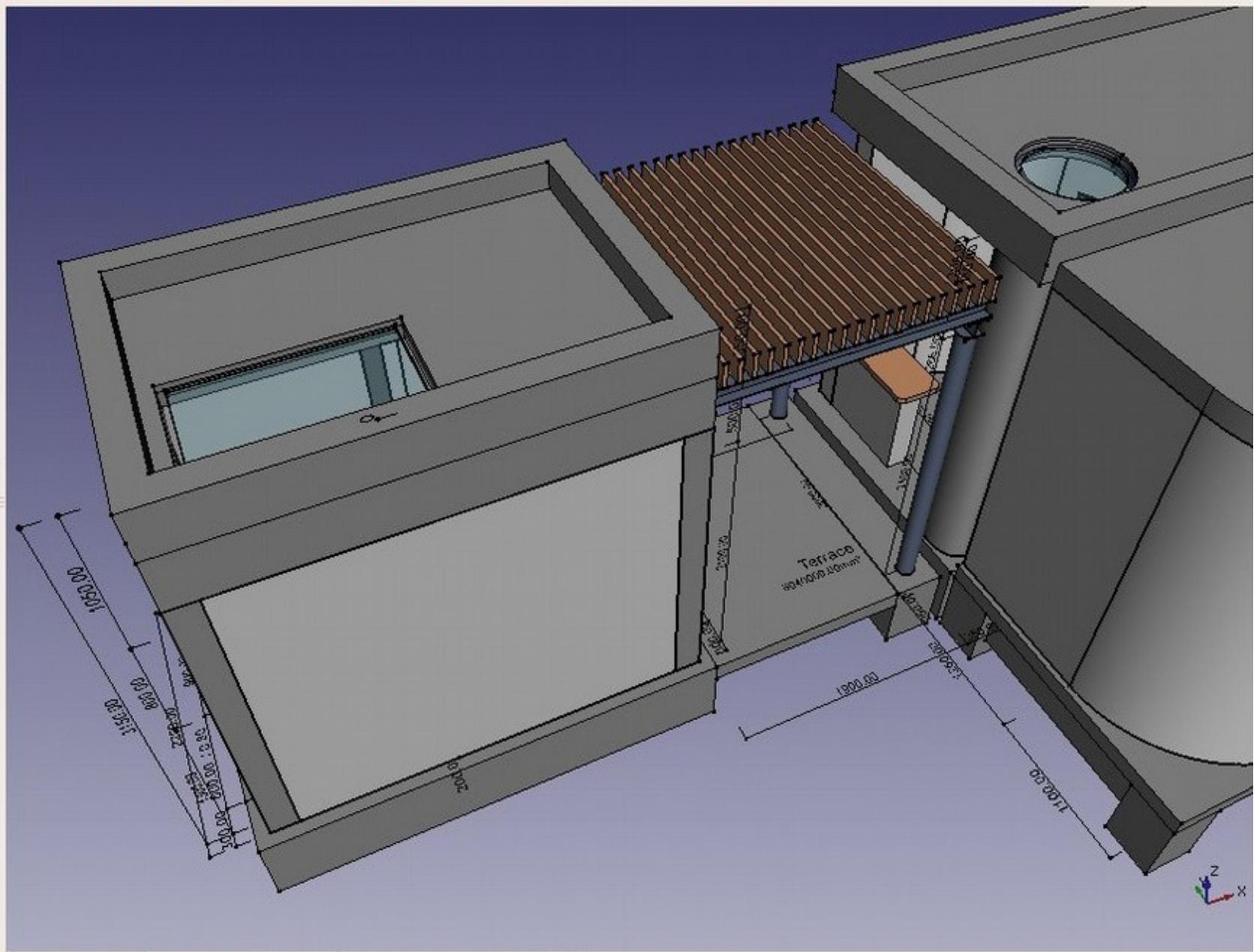
Model Tasks

Labels & Attributes

Application

- testhouse
 - Construction
 - Annotations
 - Plan Annotations
 - Section A Annotations
 - Site
 - Testhouse 2
 - Ground floor
 - Structures
 - Walls
 - Wall Short 1
 - Wall Long 1
 - Wall Short 2

Property	Value
----------	-------





Combo View

Model Tasks

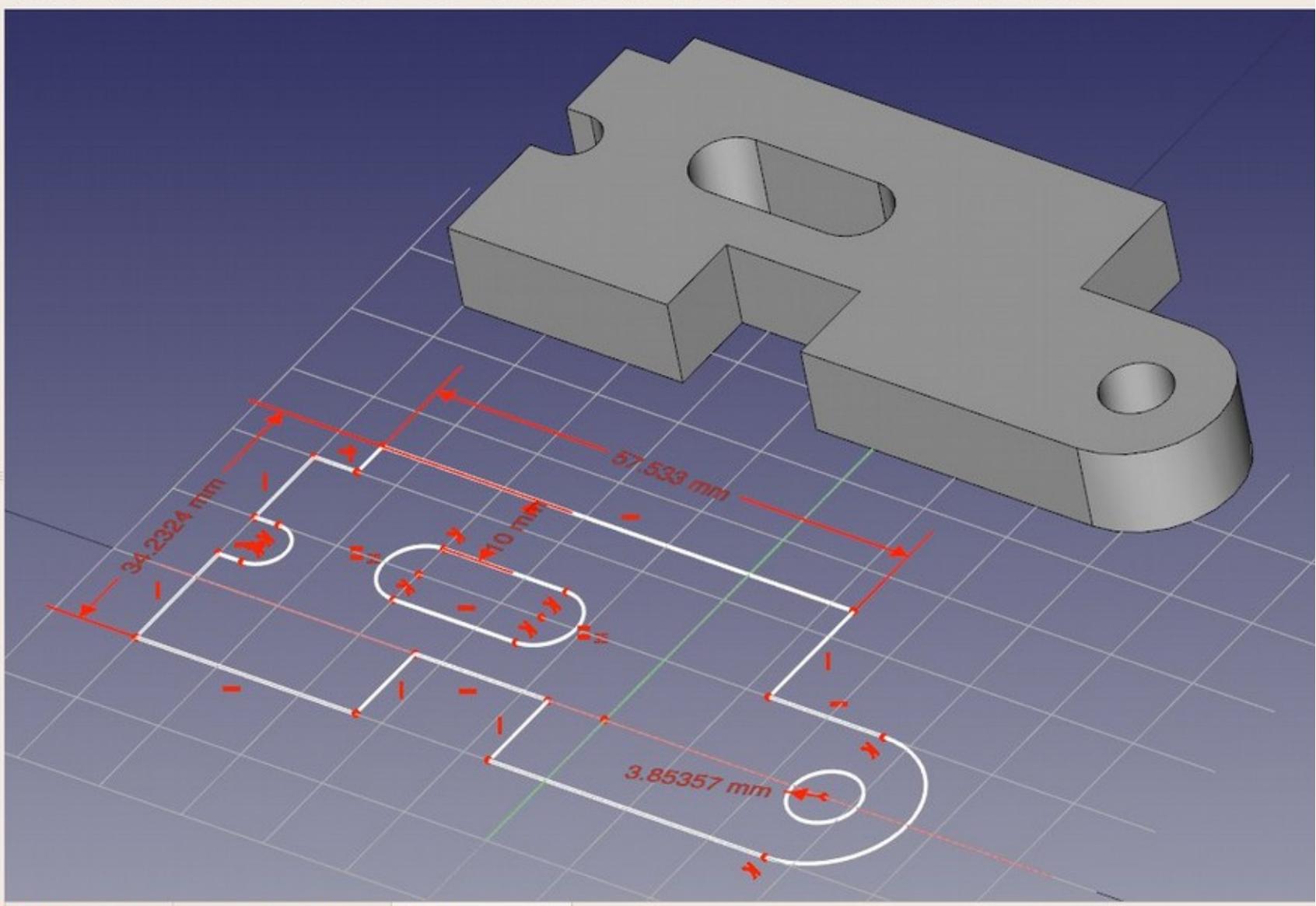
Help Close

Solver messages
Under-constrained sketch with 19 degrees of freedom
Solved in 0 sec

Edit controls
Show grid
Grid size: 10 mm
Grid snap
Auto constraints

Constraints
Filter: Normal
Constraint1
Constraint3
Constraint5
Constraint6
Constraint7
Constraint9
Constraint11

Elements
Type: Edge



testhouse : 1 testhouse : A3 Page Unnamed1 : 1*

Selection view

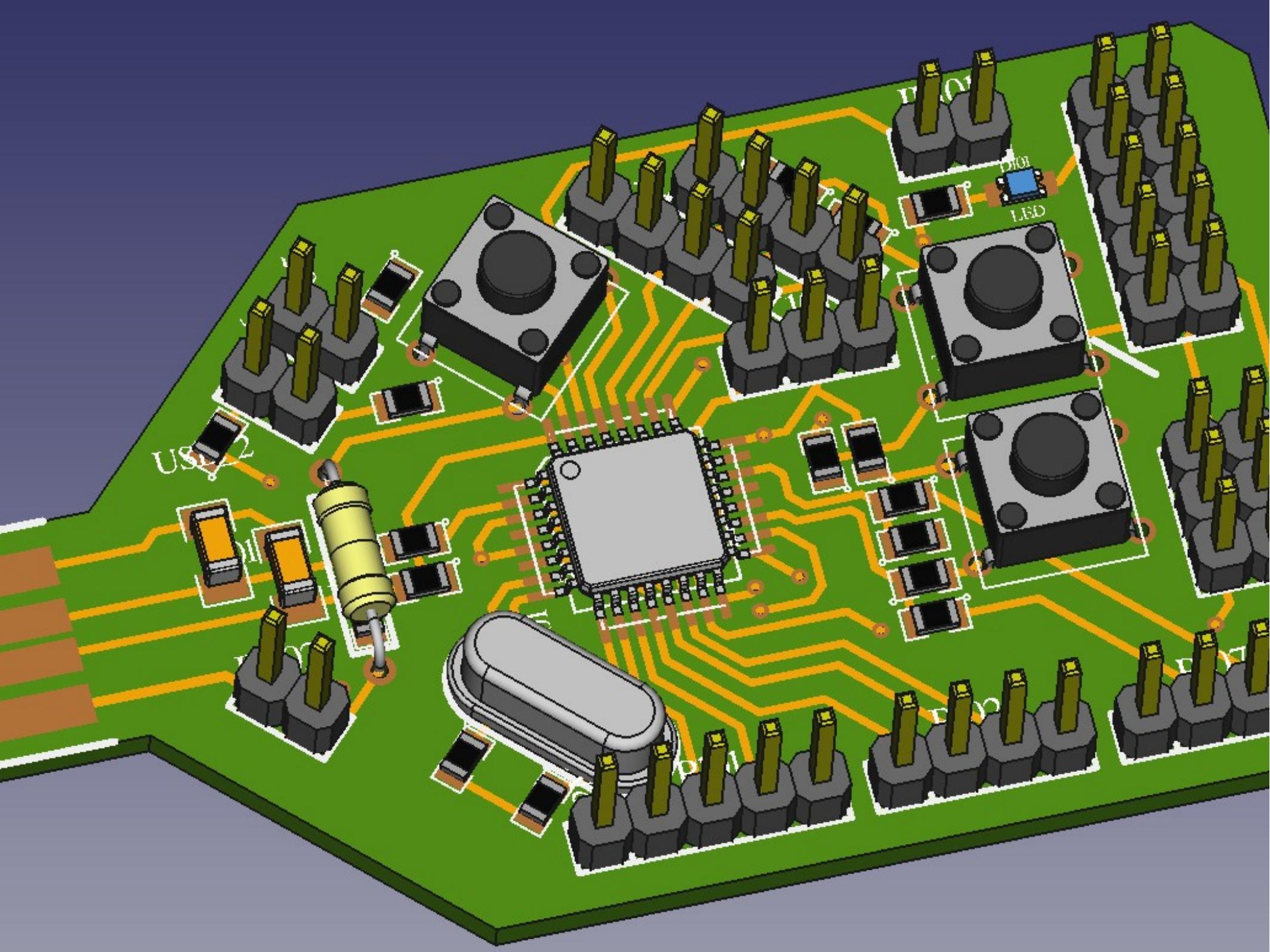
Search

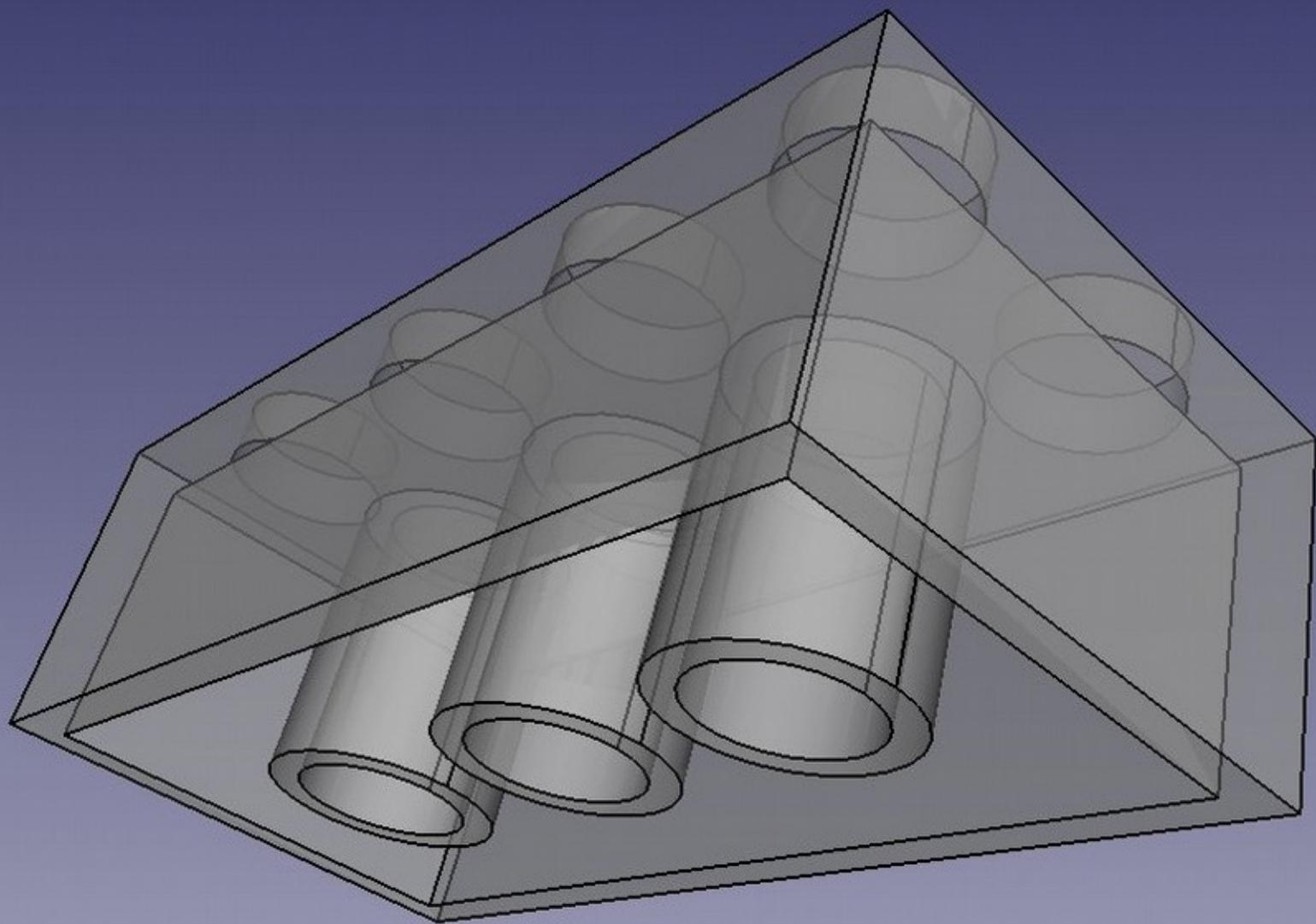
Report view

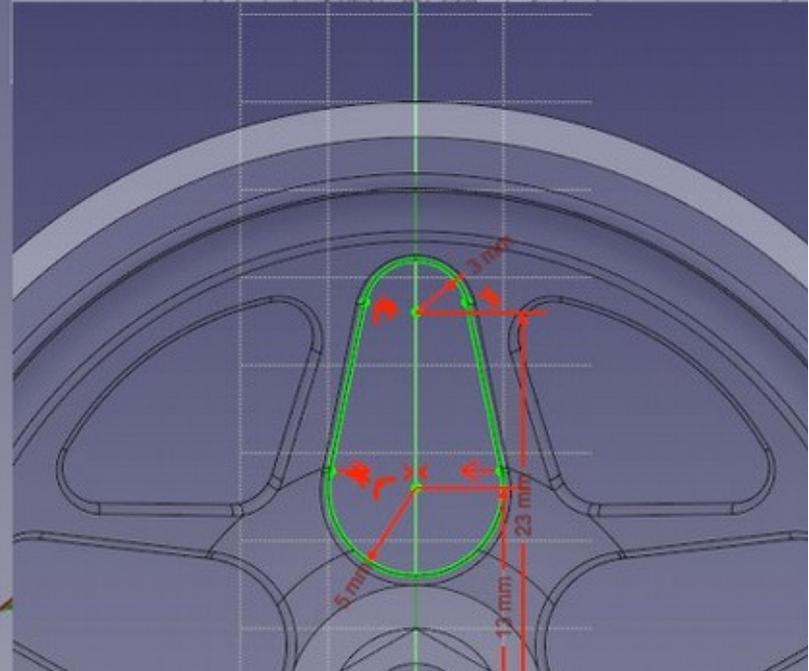
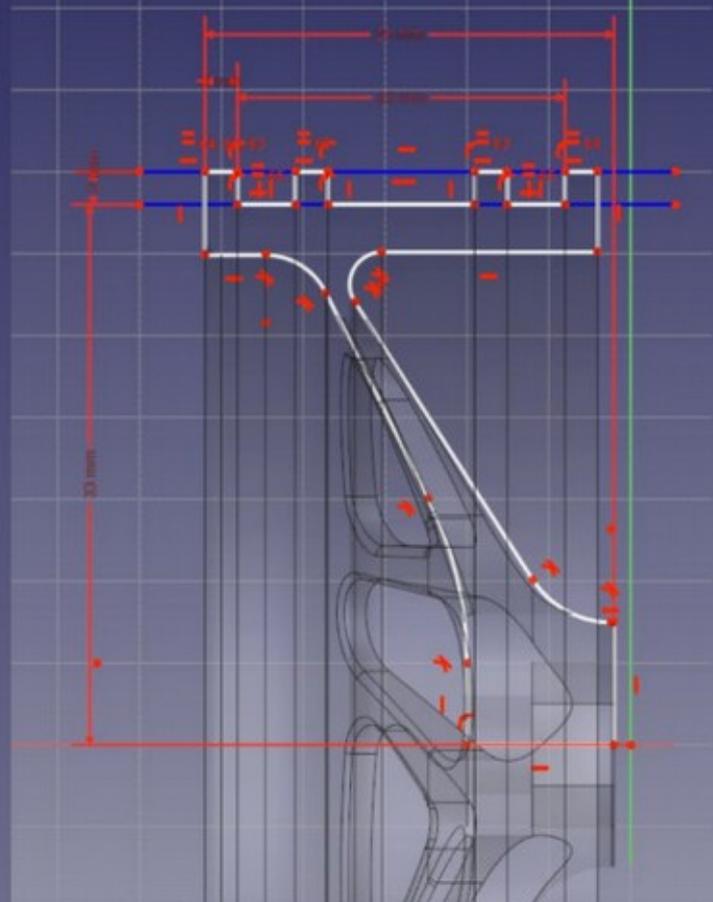
```
Sel : Add Selection "Unnamed.Sketch.(null) (0.000000,0.000000,0.000000)"  
Sel : Clear selection  
Sel : Add Selection  
"Unnamed.Pad001.Face23(-37.032871,52.429699,10.000000)"  
Active view is Unnamed1 : 1[*] (at 0x8db6980)  
Sel : Clear selection  
Active view is Unnamed1 : 1[*] (at 0x8db6980)  
Sel : Clear selection
```

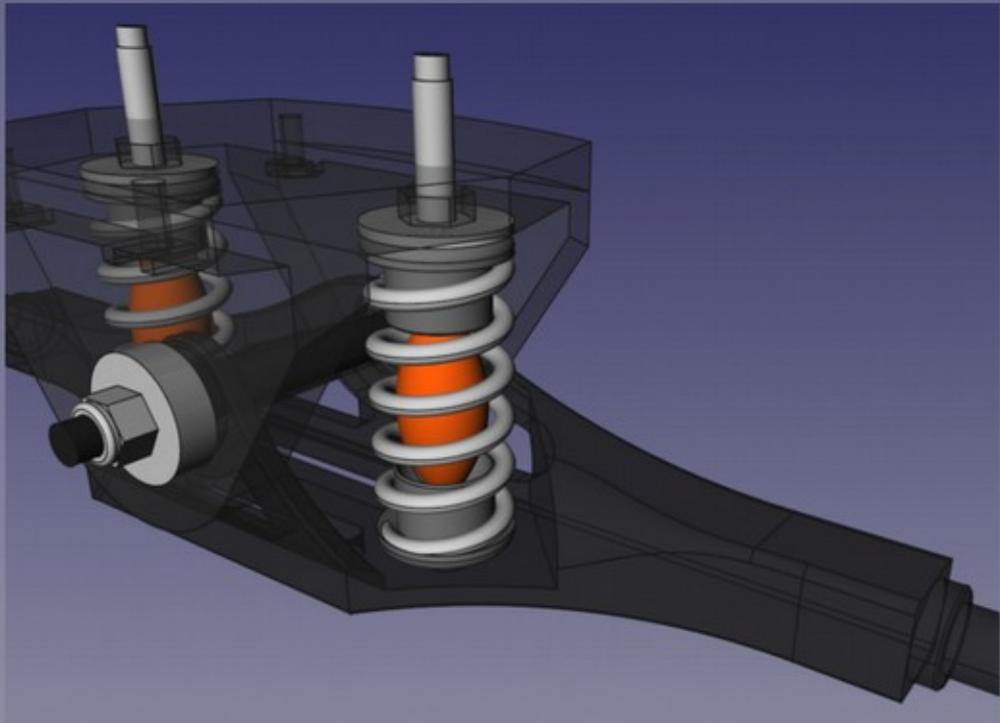
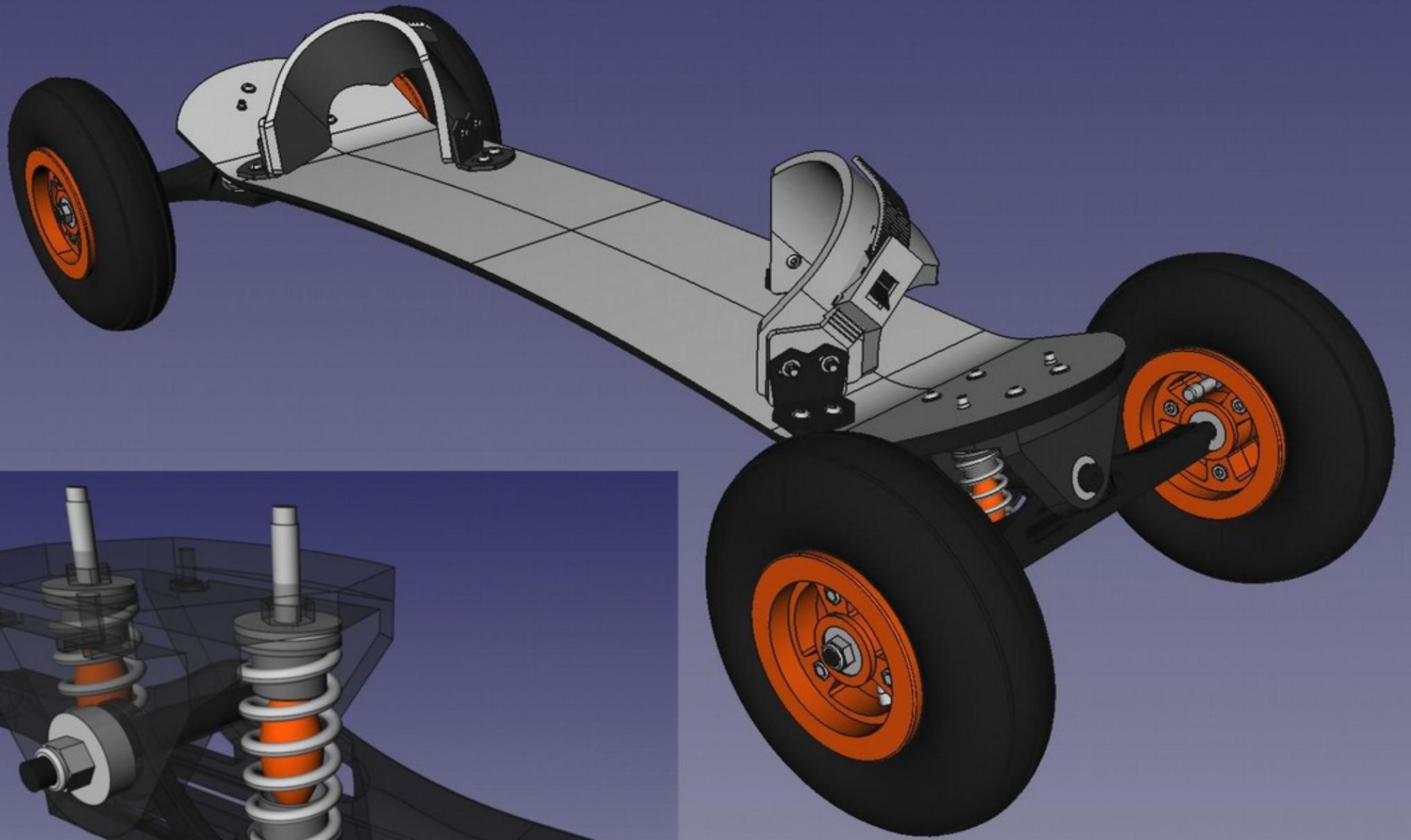
Python console

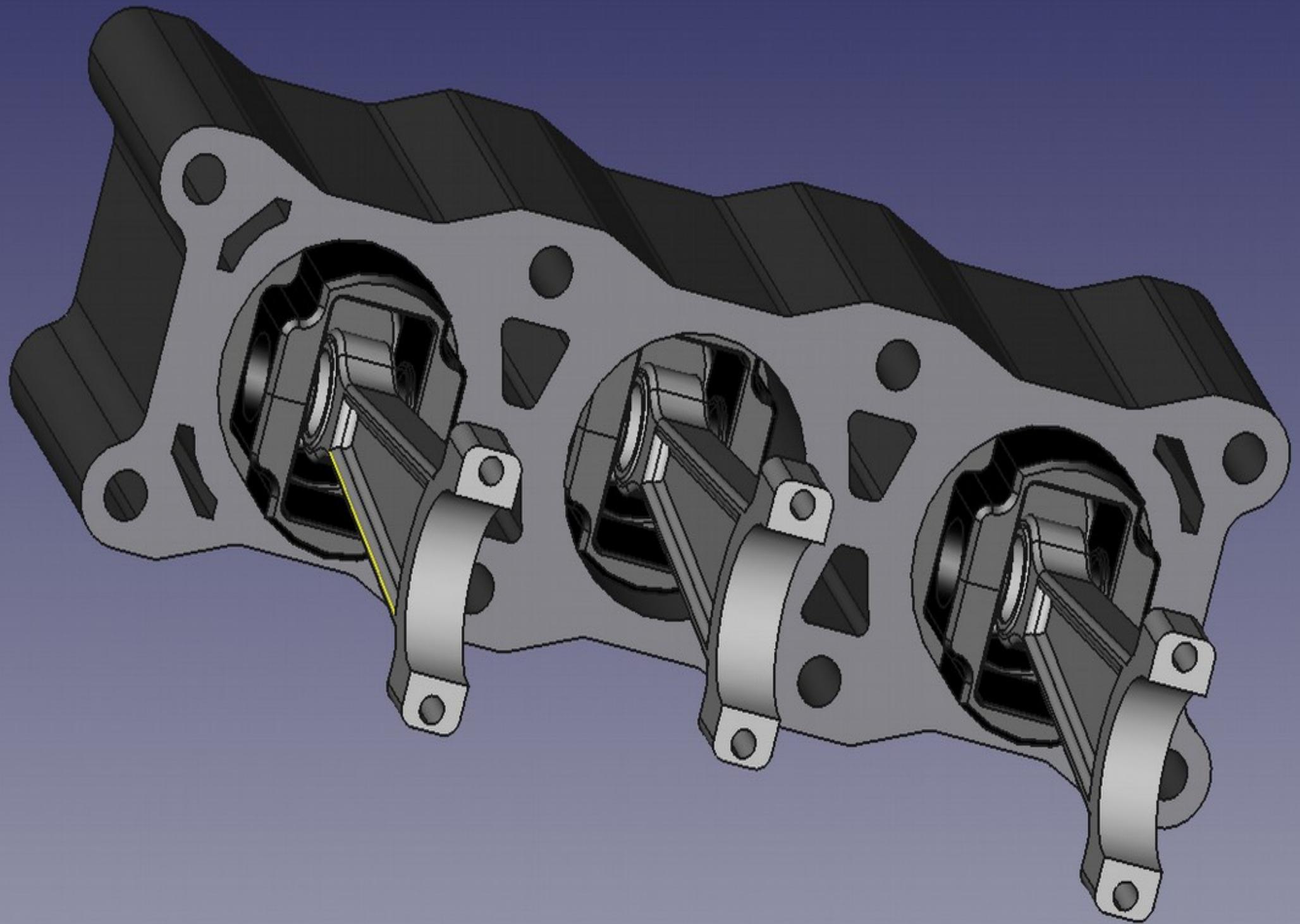
```
>>> App.getDocument('Unnamed').recompute()  
>>> Draft.move([FreeCAD.ActiveDocument.Pad001],FreeCAD.Vector(0.0,94.18  
>>> FreeCAD.ActiveDocument.recompute()  
>>> Gui.getDocument('Unnamed').resetEdit()  
>>> App.getDocument('Unnamed').recompute()  
>>> FreeCADGui.getDocument("Unnamed").getObject("Pad001").ShapeColor =  
>>> FreeCADGui.getDocument("Unnamed").getObject("Pad001").ShapeColor =  
>>>
```

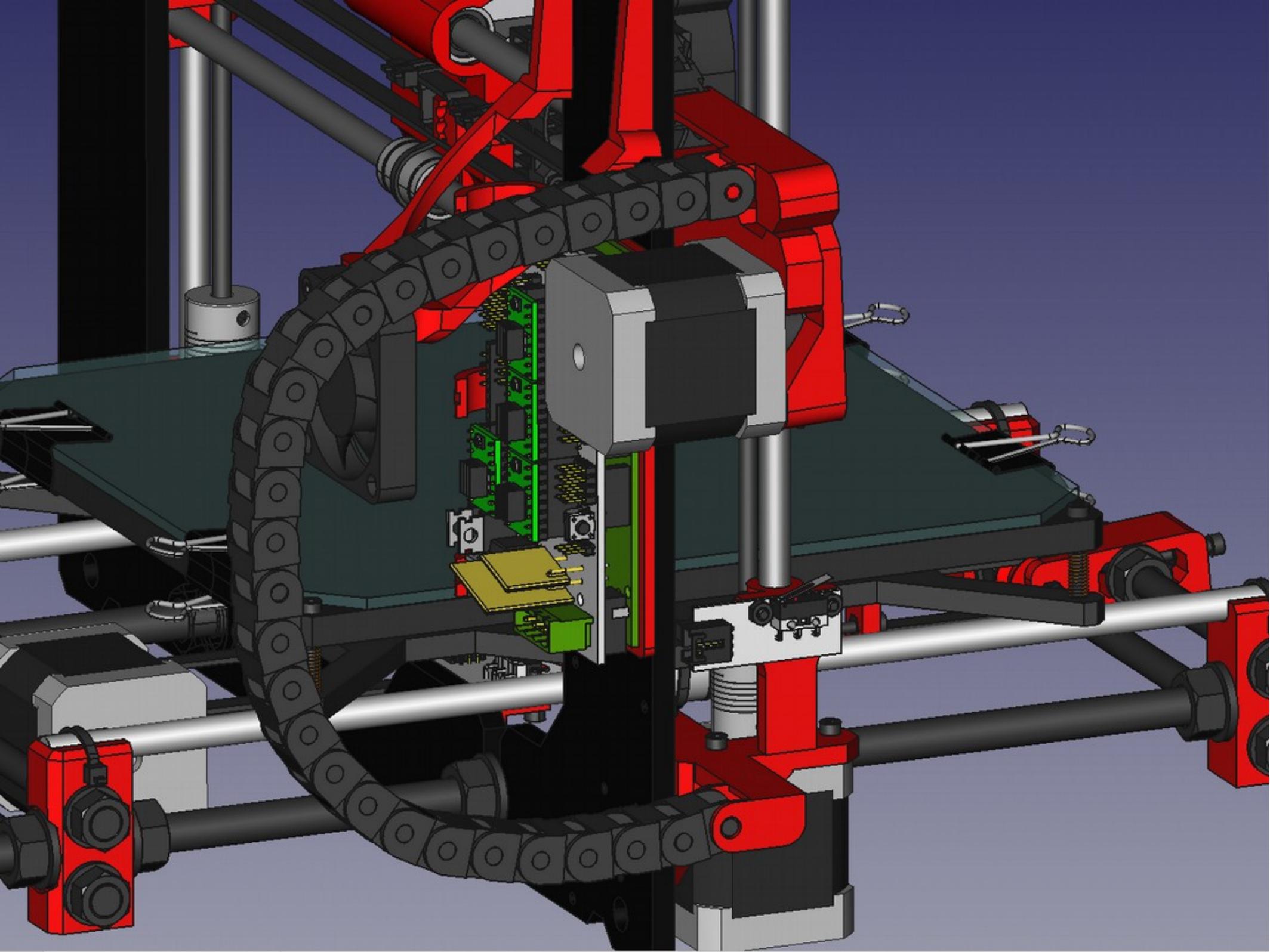


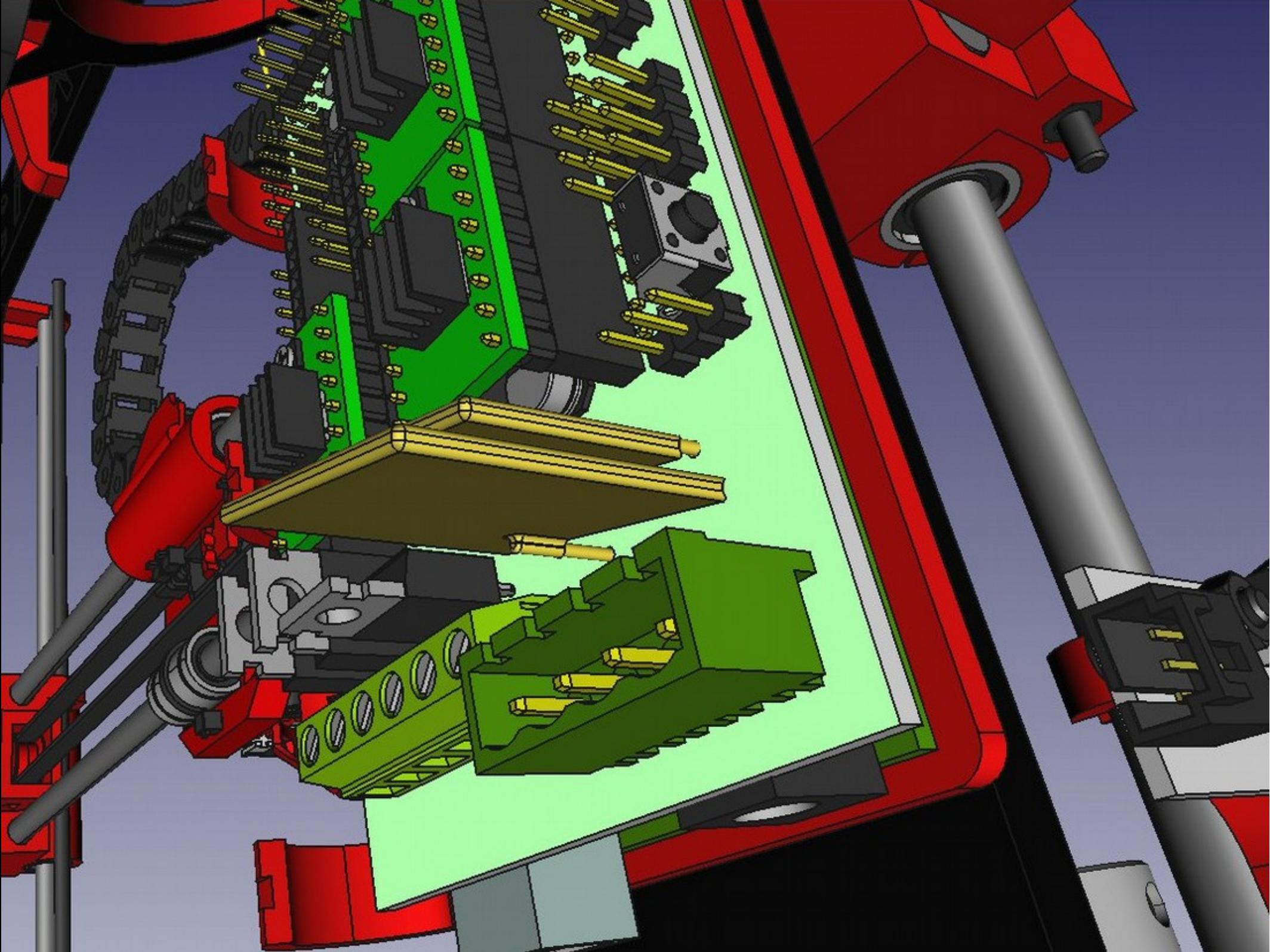


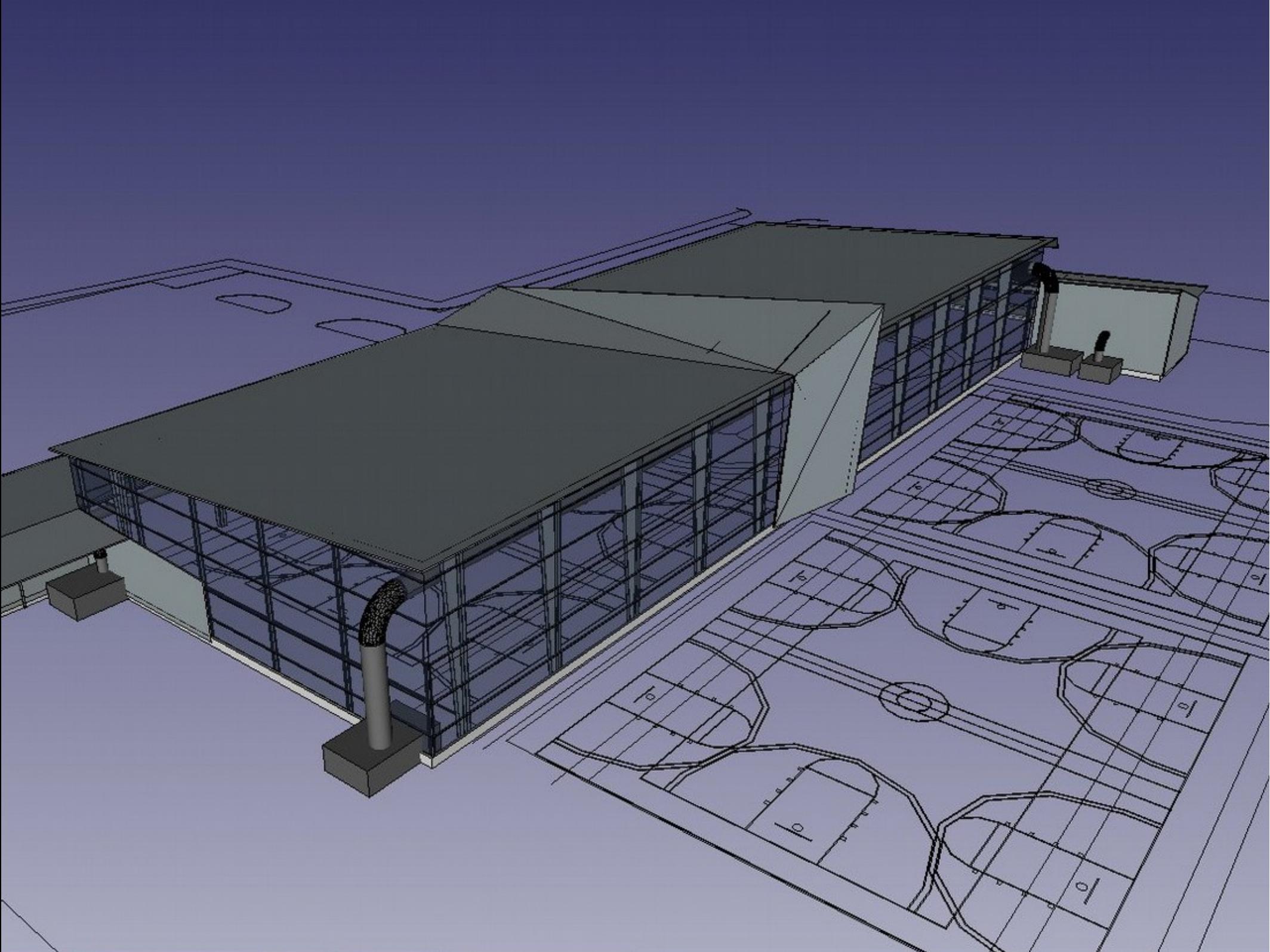


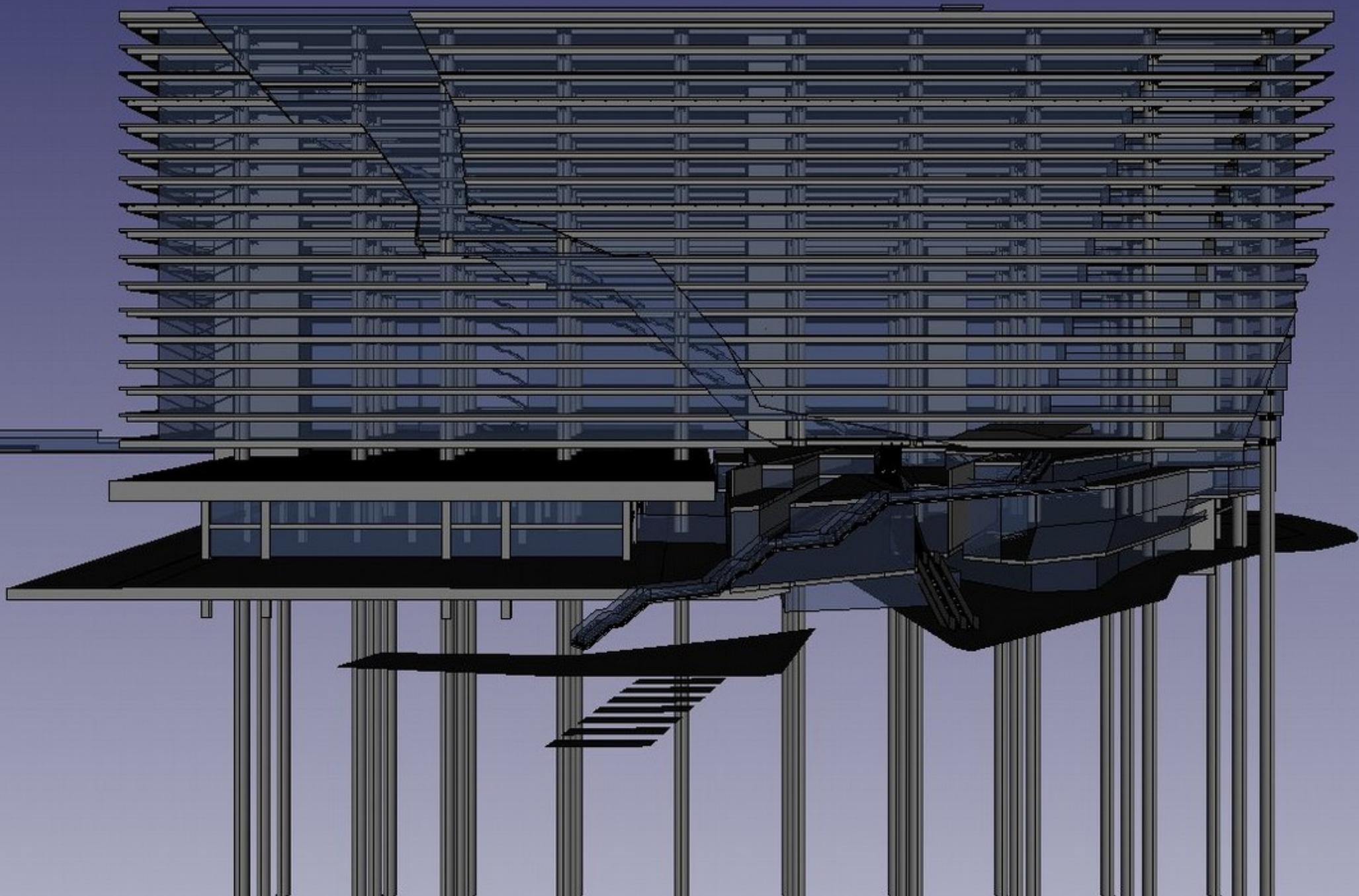


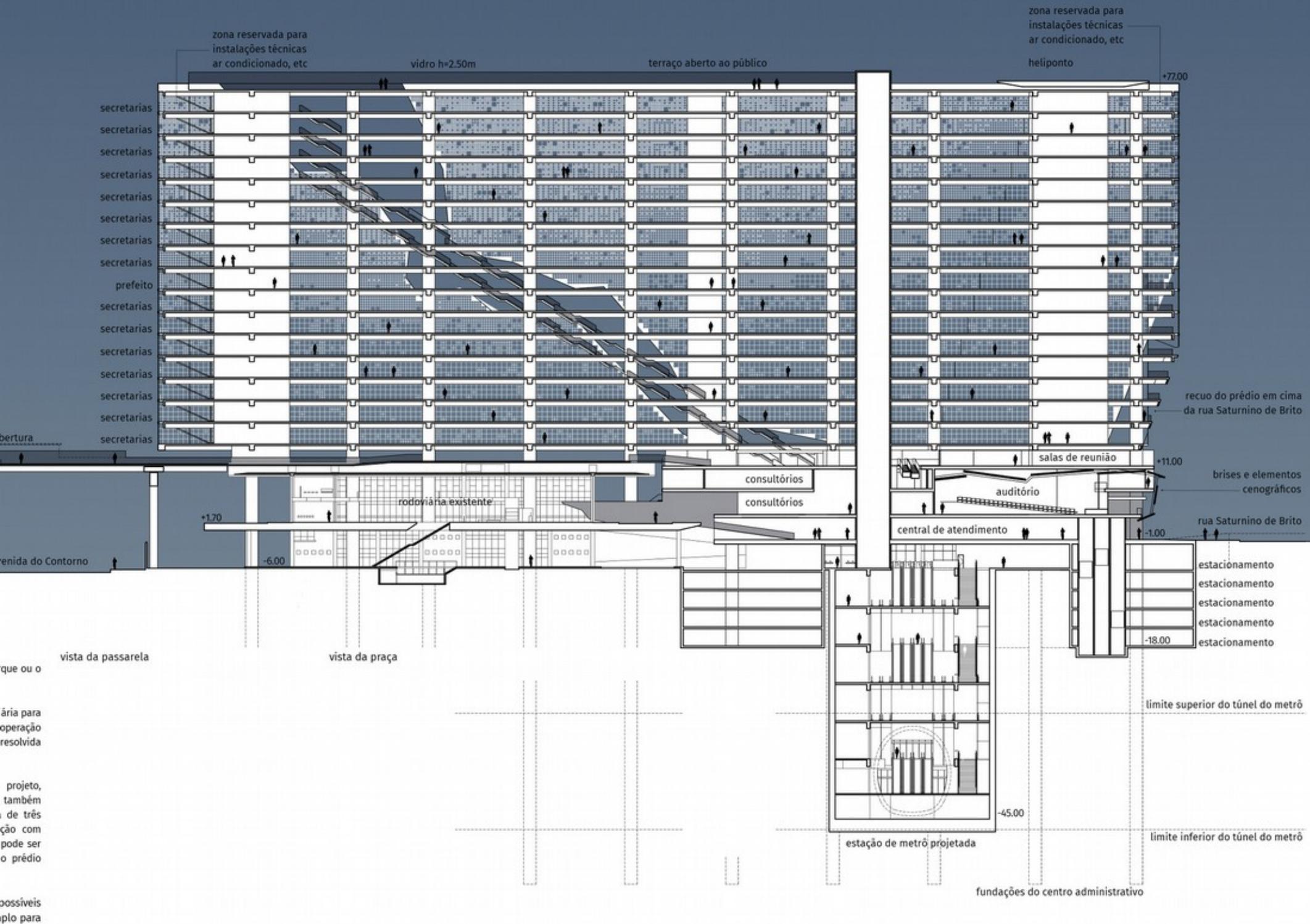












zona reservada para instalações técnicas ar condicionado, etc

vidro h=2.50m

terrace aberto ao público

zona reservada para instalações técnicas ar condicionado, etc

heliponto

+77.00

secretarias
secretarias

prefeito

abertura

+1.70

rodoviária existente

consultórios

consultórios

salas de reunião

+11.00

auditorio

central de atendimento

-1.00

recuo do prédio em cima da rua Saturnino de Brito

brises e elementos cenográficos

rua Saturnino de Brito

estacionamento
estacionamento
estacionamento
estacionamento
estacionamento

limite superior do túnel do metrô

-45.00

estação de metrô projetada

limite inferior do túnel do metrô

fundações do centro administrativo

vista da passarela

vista da praça

que ou o

ária para
operação
resolvida

projeto,
também
de três
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pode ser
o prédio

possíveis
mplo para

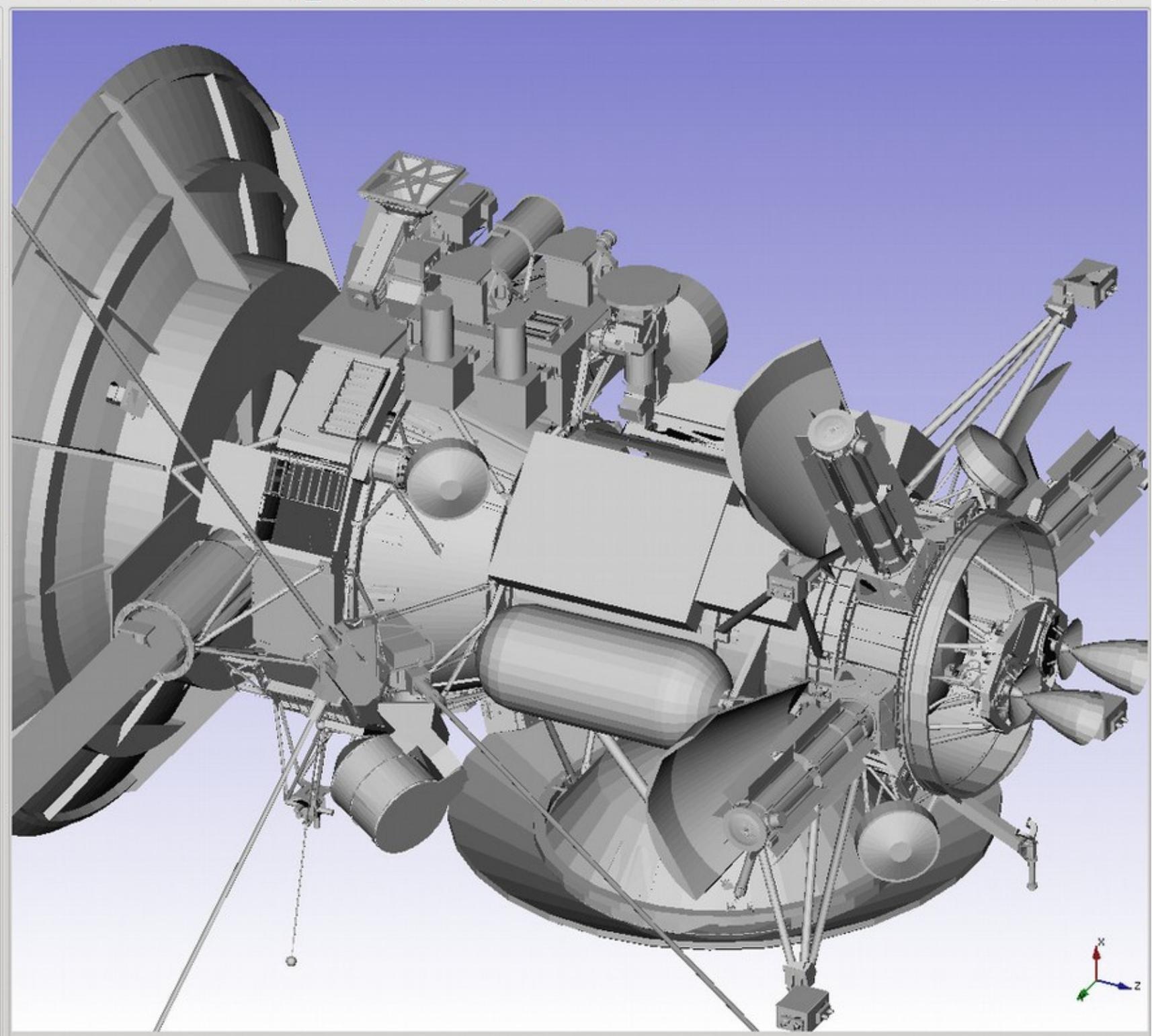


Combo View

Project

- Labels_Attributes
- Application
 - Unnamed
 - cassini

Property	Value
Base	
Display M...	File+Buffer
Visibility	true



Bancadas genéricas

- Draft: ferramentas genéricas 2D (linha, retângulo, etc) e modificação geral (deslocamento, rotação, etc)
- Part: ferramentas genéricas 3D (cubo, esfera, etc) e operações booleanas (união, subtração, interseção)



Combo View

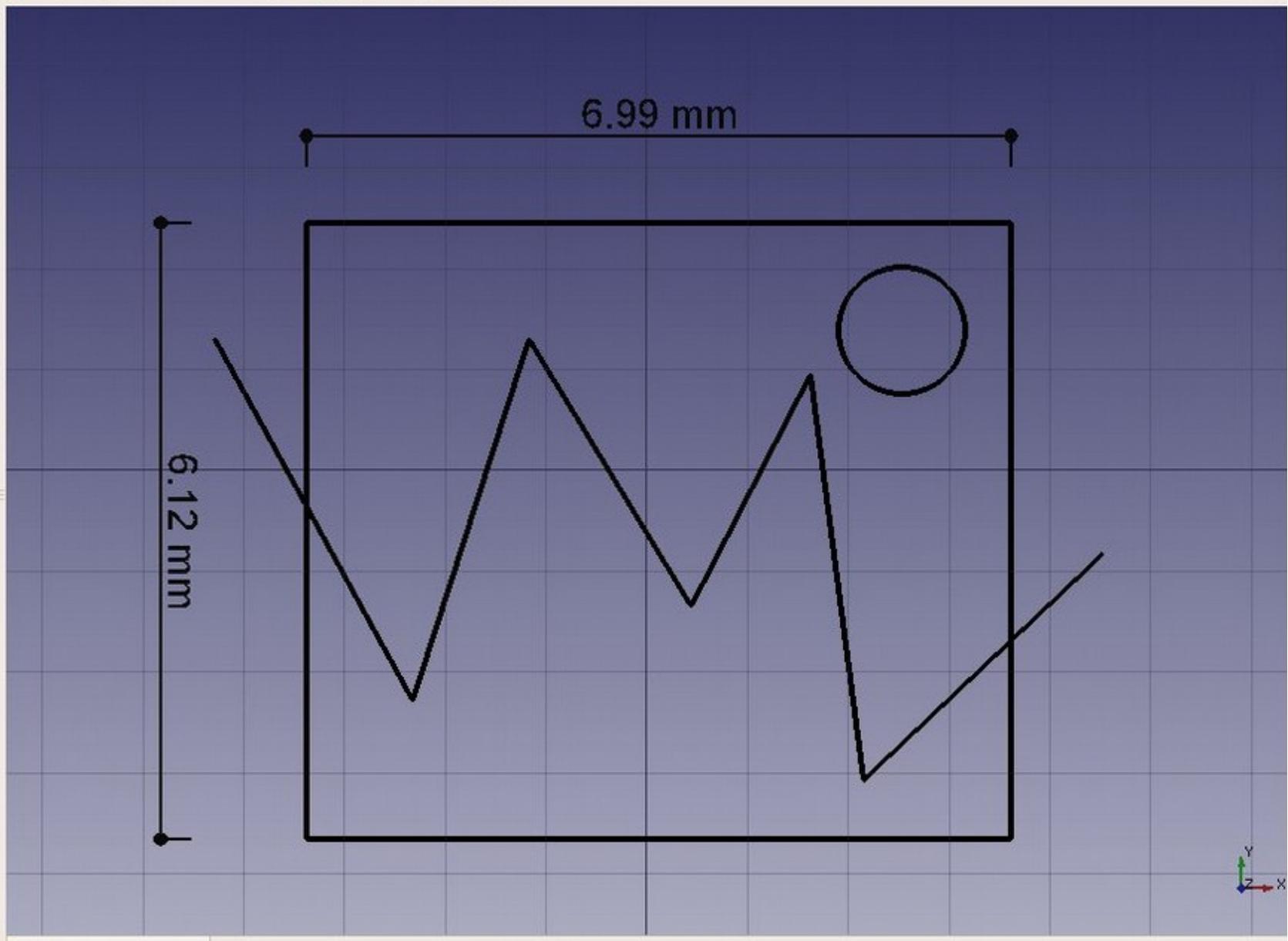
Model Tasks

Labels & Attributes

Application

- ▼ Unnamed
 - Rectangle
 - DWire
 - Circle
 - Dimension
 - Dimension001

Property	Value
----------	-------





Combo View  

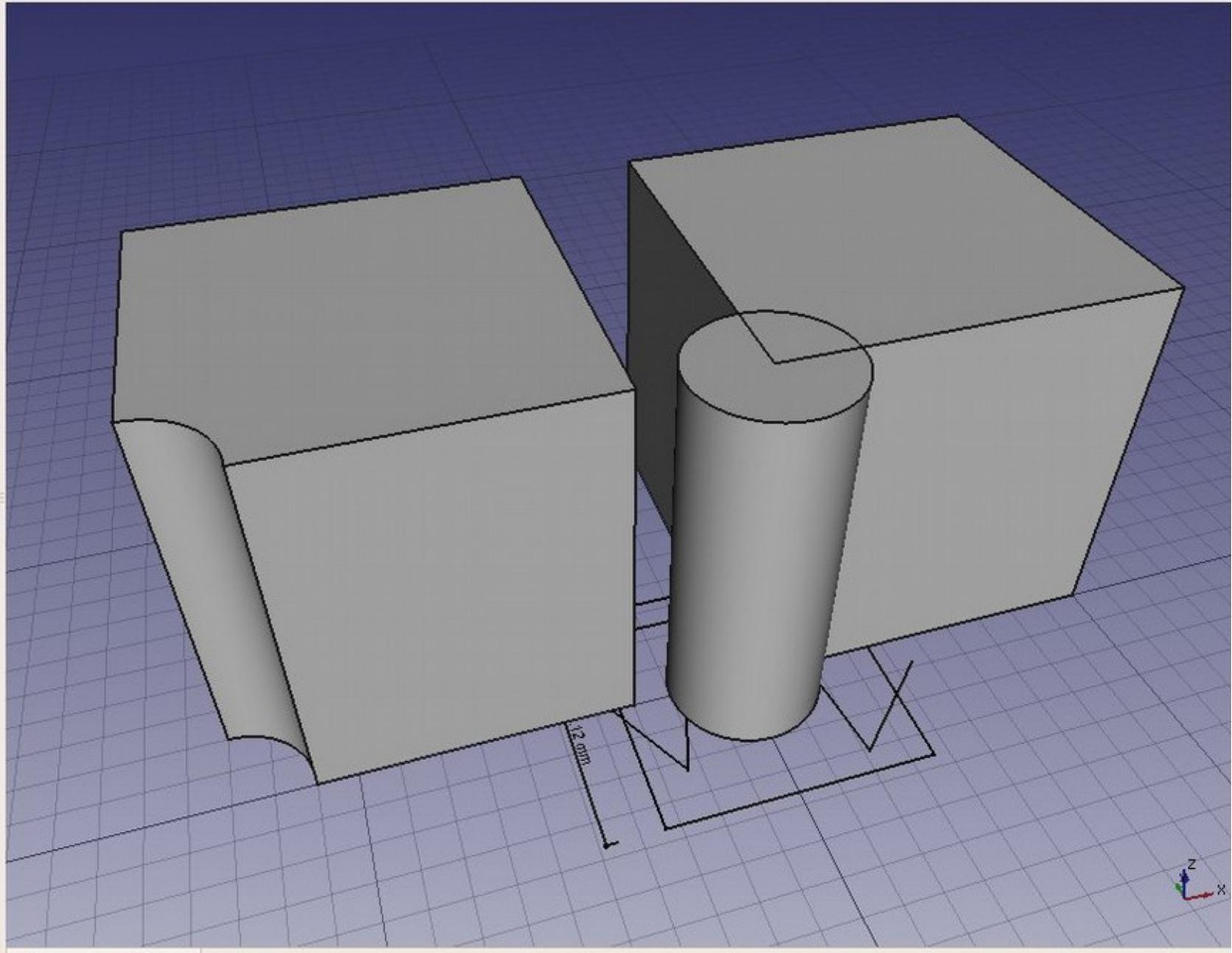
Model Tasks

Labels & Attributes

Application

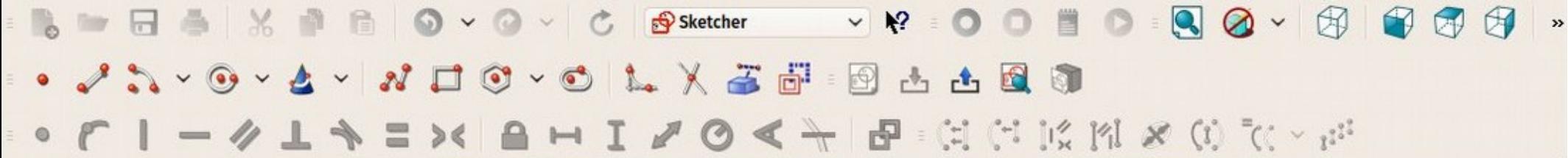
- ▼ Unnamed
 - Rectangle
 - DWire
 - Circle
 - Dimension
 - Dimension001
 - Cube
 - Cylinder
 - ▼ Cut
 - Cube001
 - Cylinder001

Property	Value
Base	
► Placement	[(0.00 0.00 1.00); 0 °...
Label	Cube001
Box	
Length	10 mm
Width	10 mm
Height	10 mm



Design para impressão 3D

- Sketcher : desenho 2D paramétrico (com restrições: comprimento, ângulo, distância, etc)
- Design de peças: ferramentas de modelagem 3D baseado no Sketcher, sempre produz sólidos



Combo View

Model Tasks

Close

Solver messages

Under-constrained sketch with 10 degrees of freedom
Solved in 0.001 sec

Auto Update Update

Edit controls

Constraints

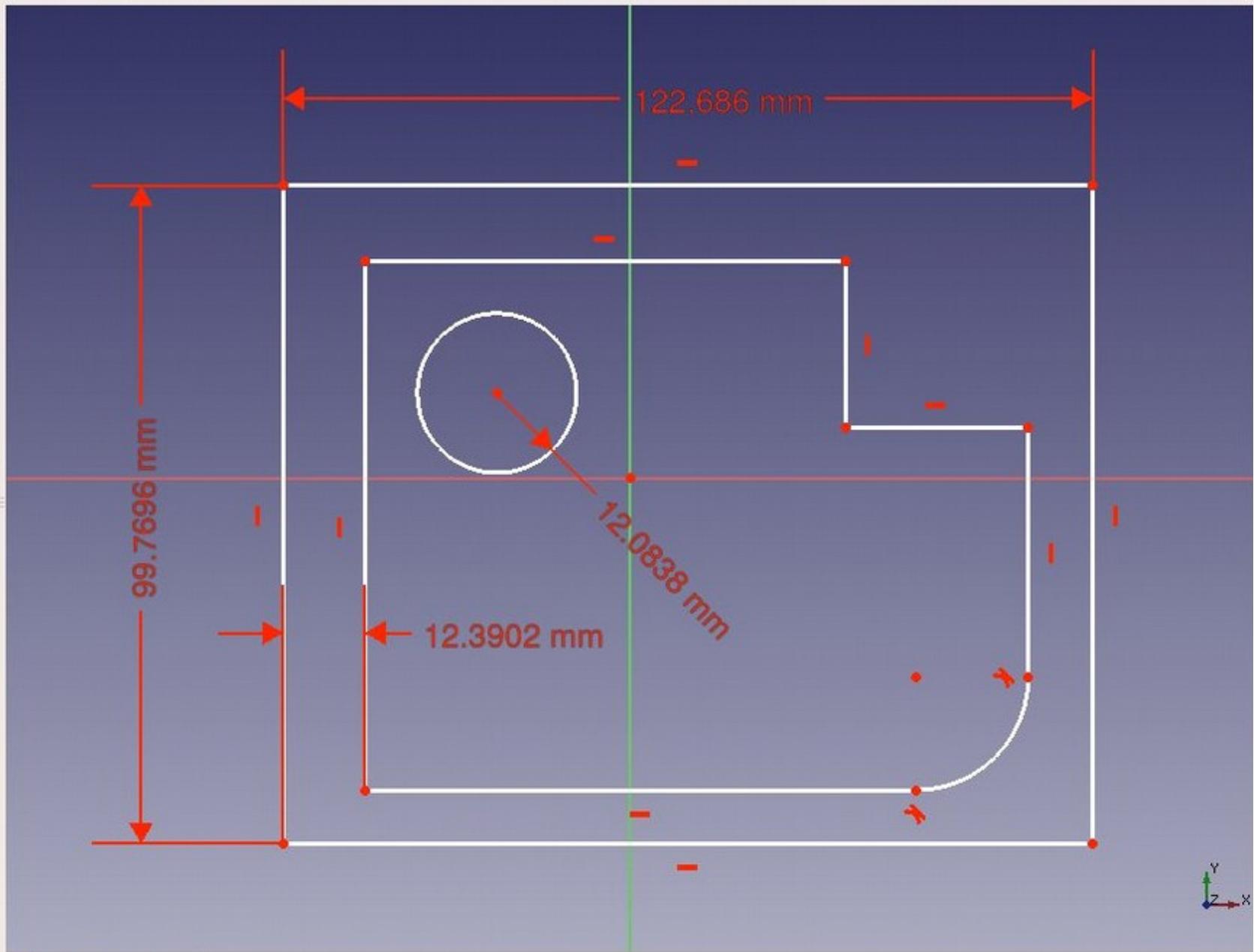
Filter: All

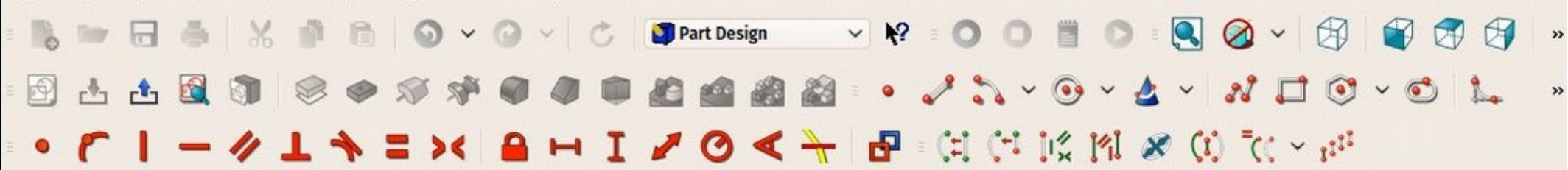
- Constraint1
- Constraint2
- Constraint3
- Constraint4
- Constraint5
- Constraint6
- Constraint7

Elements

Type: Edge

1-Line





Combo View

Model Tasks

Close

Solver messages

Under-constrained sketch with 4 degrees of freedom
Solved in 0 sec

Auto Update Update

Edit controls

Constraints

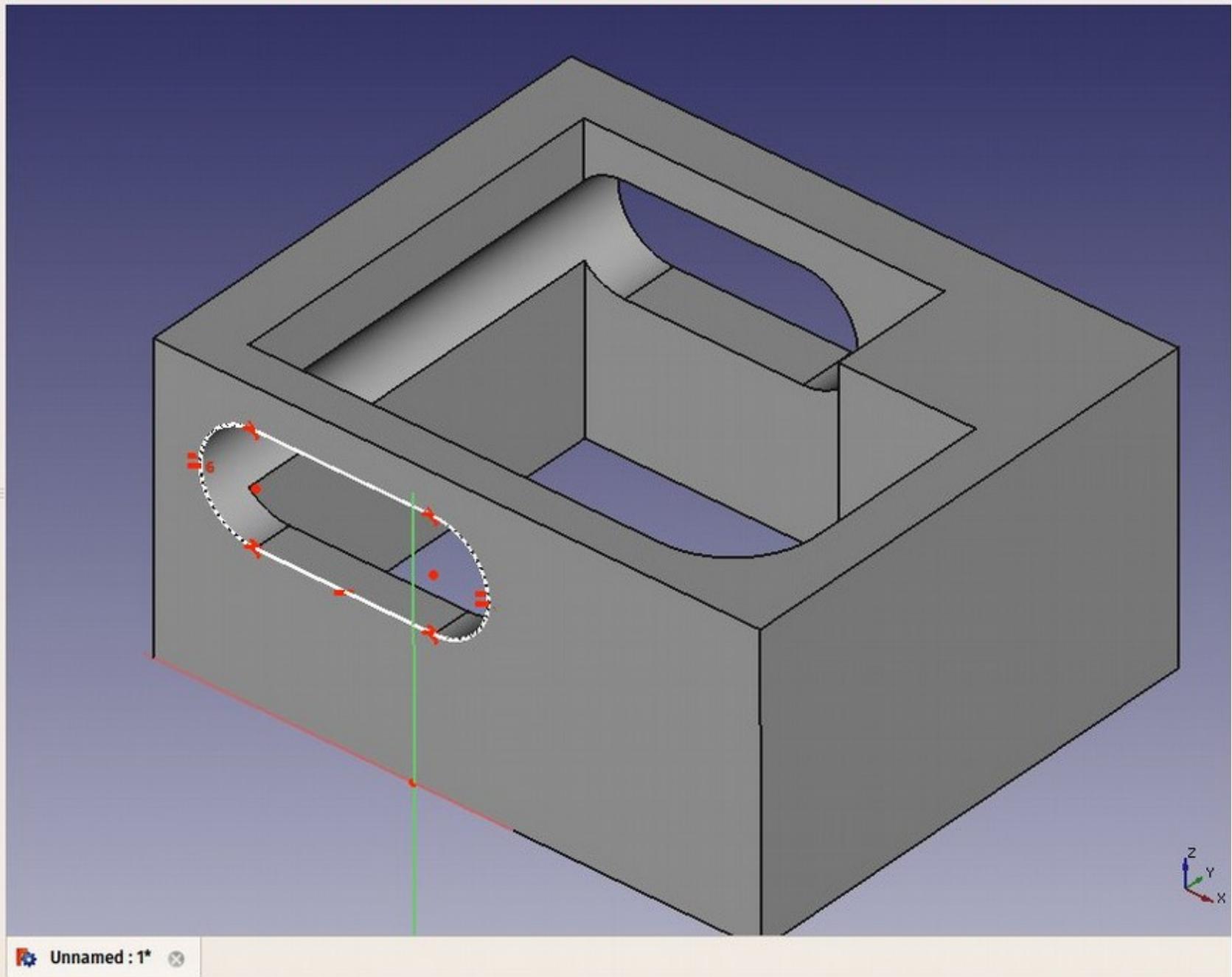
Filter: All

- Constraint1
- Constraint2
- Constraint3
- Constraint4
- Constraint5
- Constraint6

Elements

Type: Edge

1-Arc



Arquitetura e BIM

- Modelagem para BIM: objetos paramétricos com semântica correta (parede, janela, etc) e quantidades medíveis
- Diálogo com aplicações BIM comerciais (Revit, ArchiCAD, etc) via IFC, geração de desenhos 2D



Combo View

Model Tasks

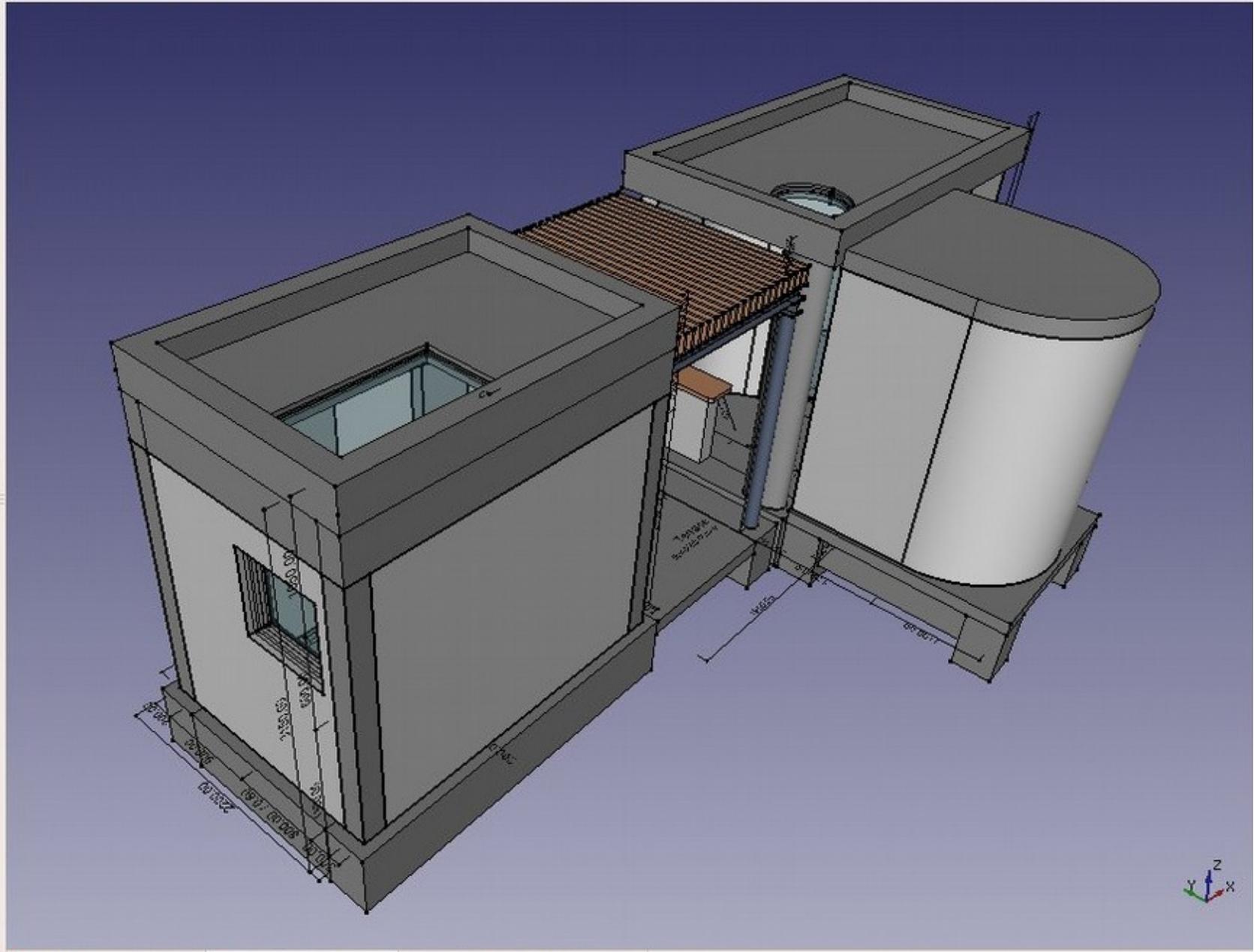
Labels & Attributes

Application

- ▼ Unnamed
 - ▶ Pad
 - ▼ Pocket
 - Sketch001
- ▼ testhouse
 - ▶ Construction
 - ▼ Annotations
 - ▶ Plan Annotations
 - ▶ Section A Annotations
 - ▼ Site
 - ▼ Testhouse 2
 - ▼ Ground floor
 - ▶ Structures

Property	Value
Line Color	[255, 255, 2...]
Line Width	2.00
Point Color	[255, 255, 2...]
Point Size	4.00
Selectable	true
Shape Color	[204, 204, 2...]
Transparency	0
Visibility	true
Grid	
Grid Size	10 mm
Grid Snap	false
Grid Style	Dashed
Show Grid	false

View Data





Combo View  

Model Tasks

Labels & Attributes

- Page title
- Project title
- Scale
- Made with FreeCAD
- Section Symbol L
- Section Symbol R
- Title Plan view
- Title section view

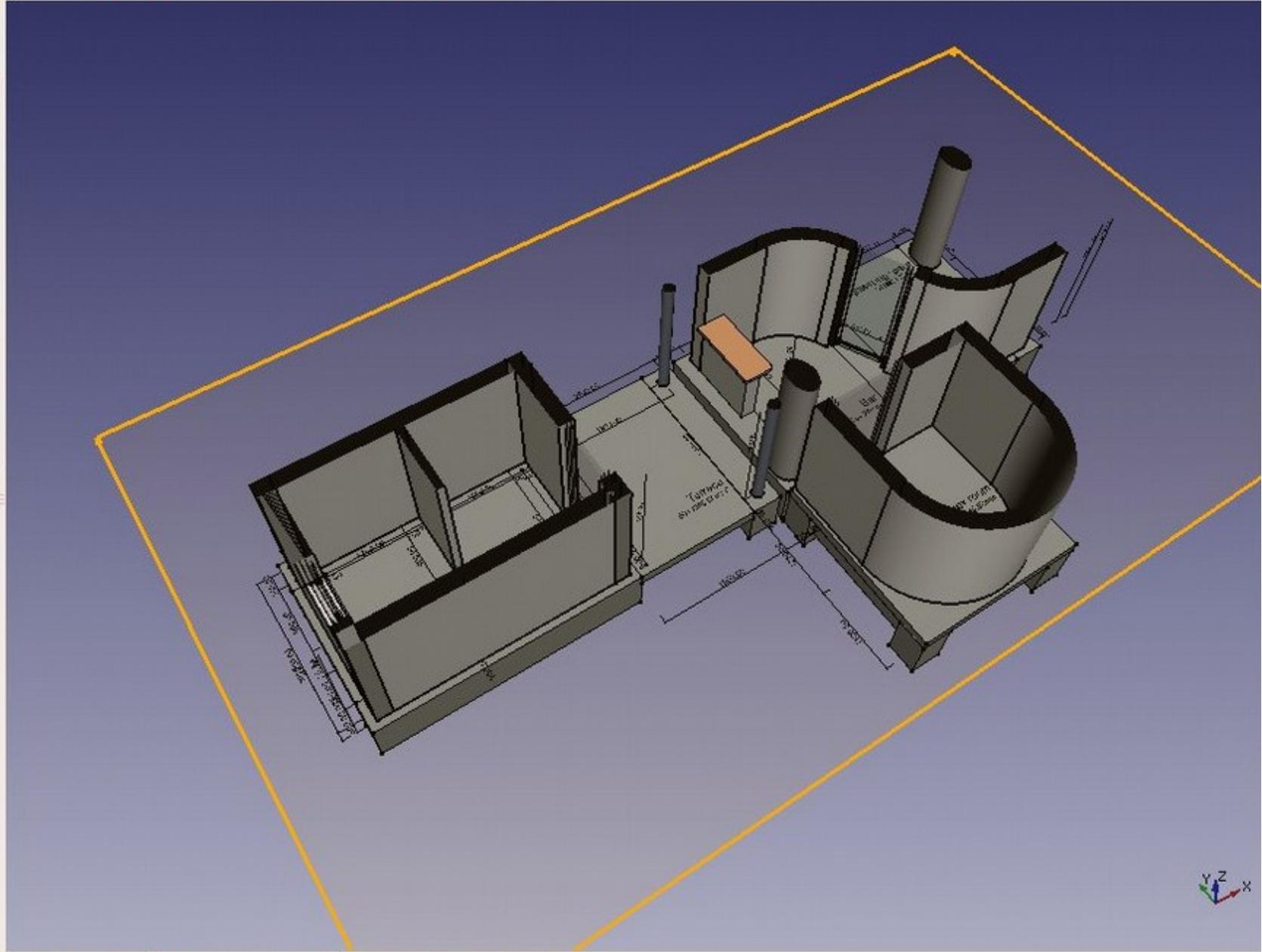
Sections

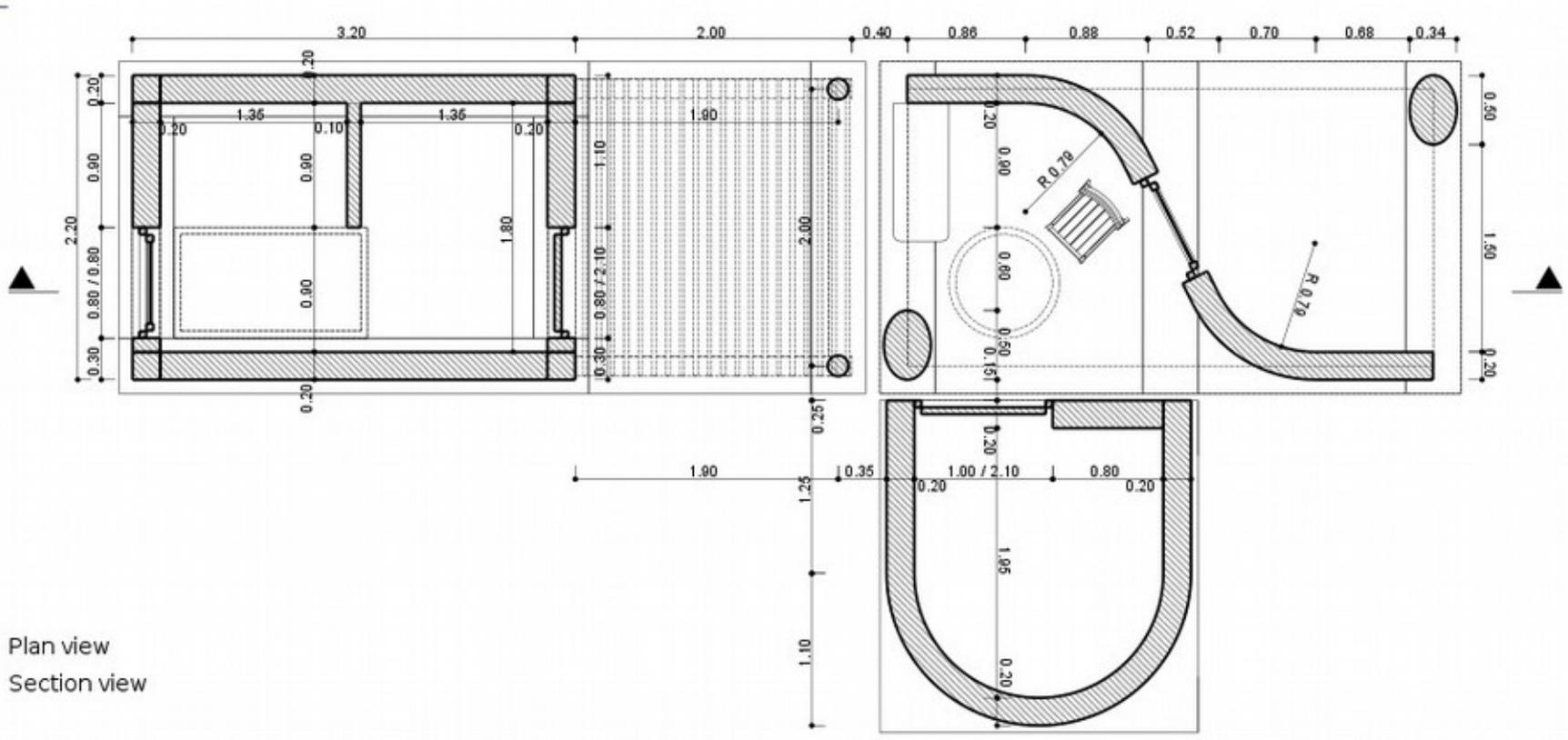
-  Section A
-  Plan

Materials

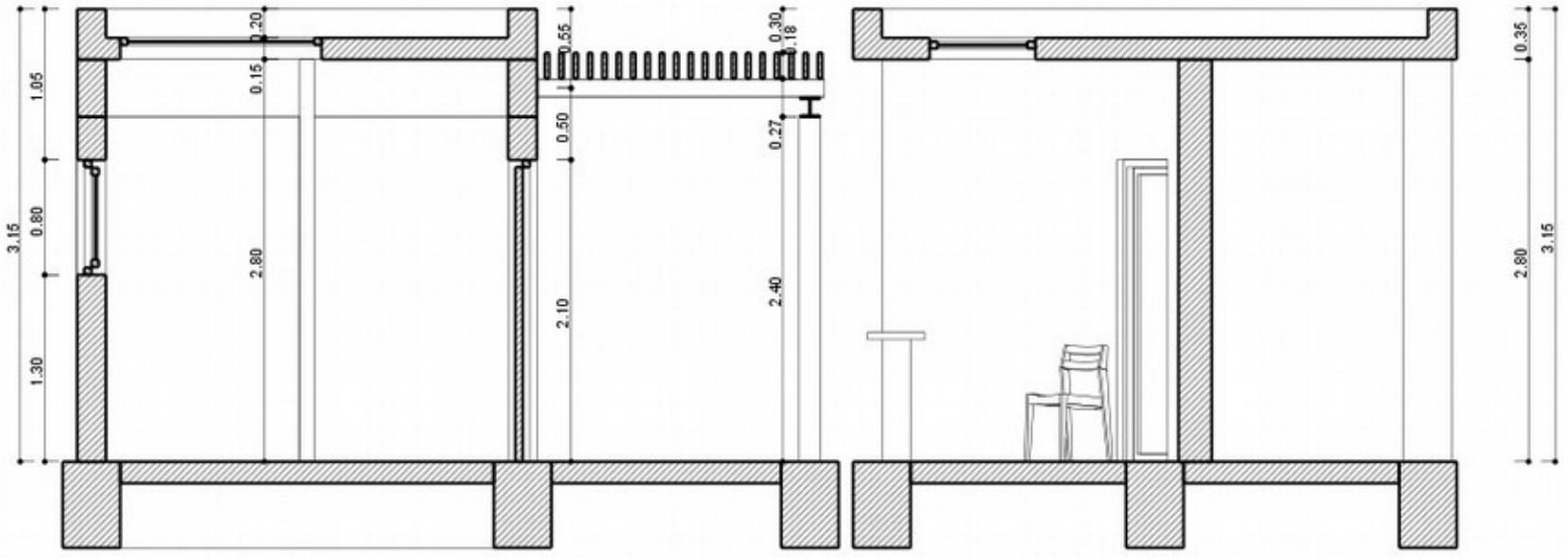
-  Quantities schedule

Property	Value
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Plan view
Section view



Test House

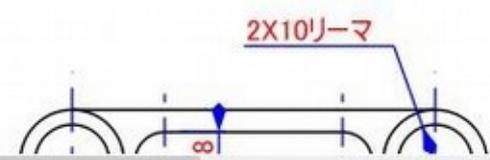
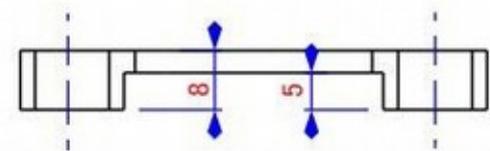
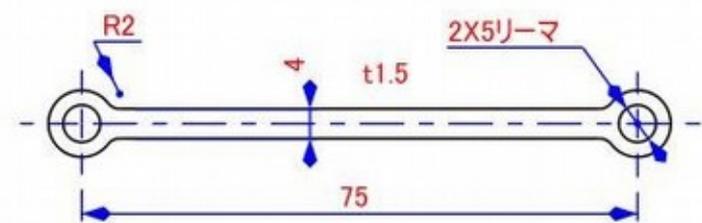
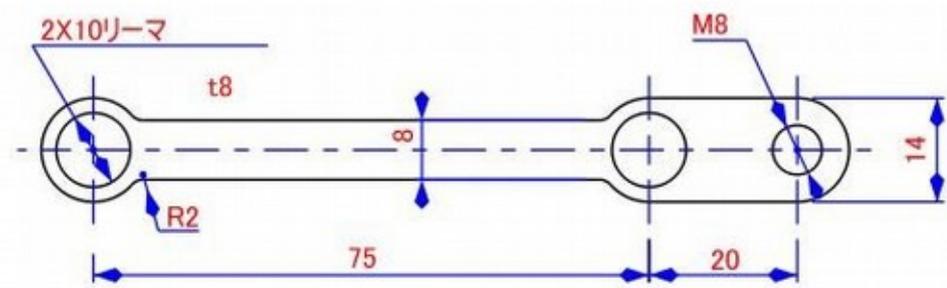
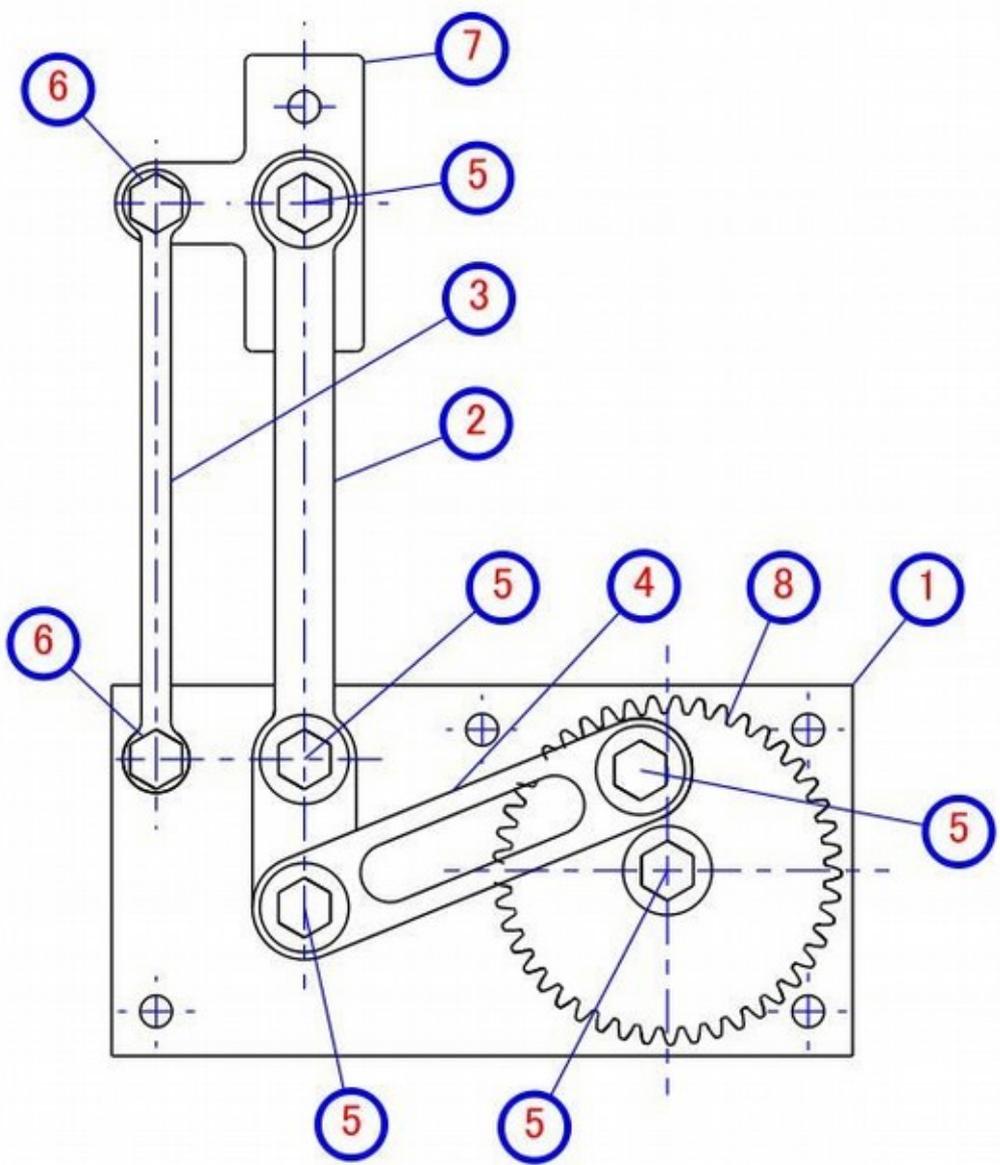
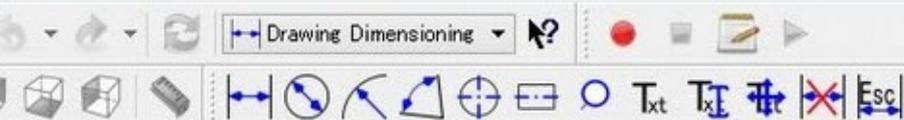
Plan and Section

Scale: 1:33



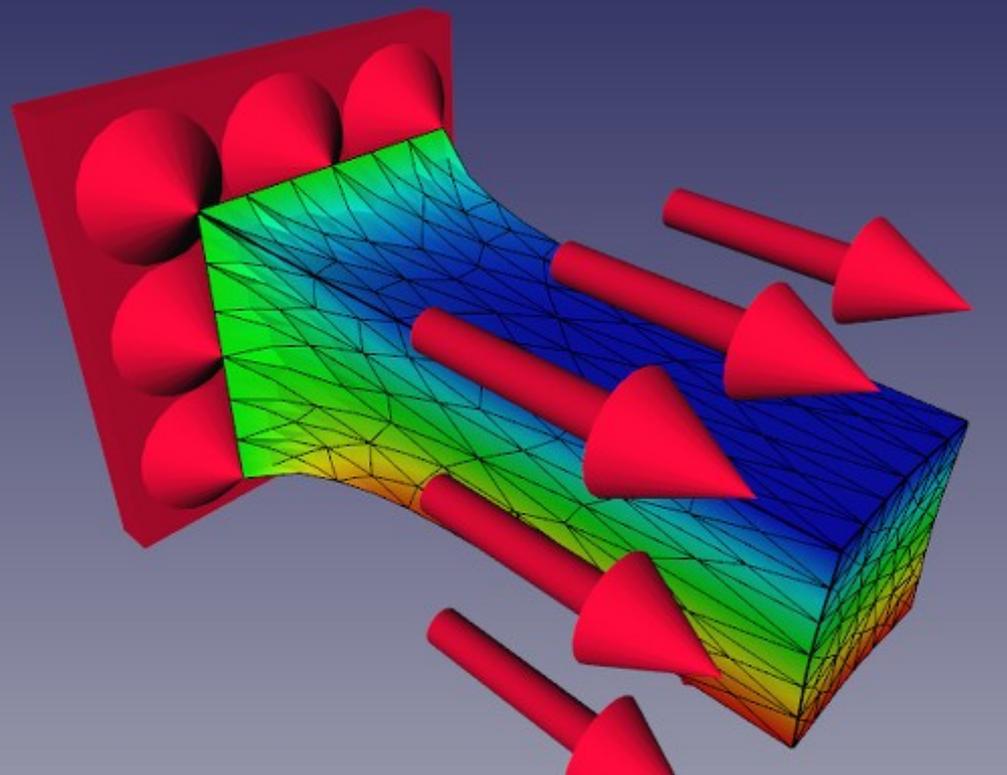
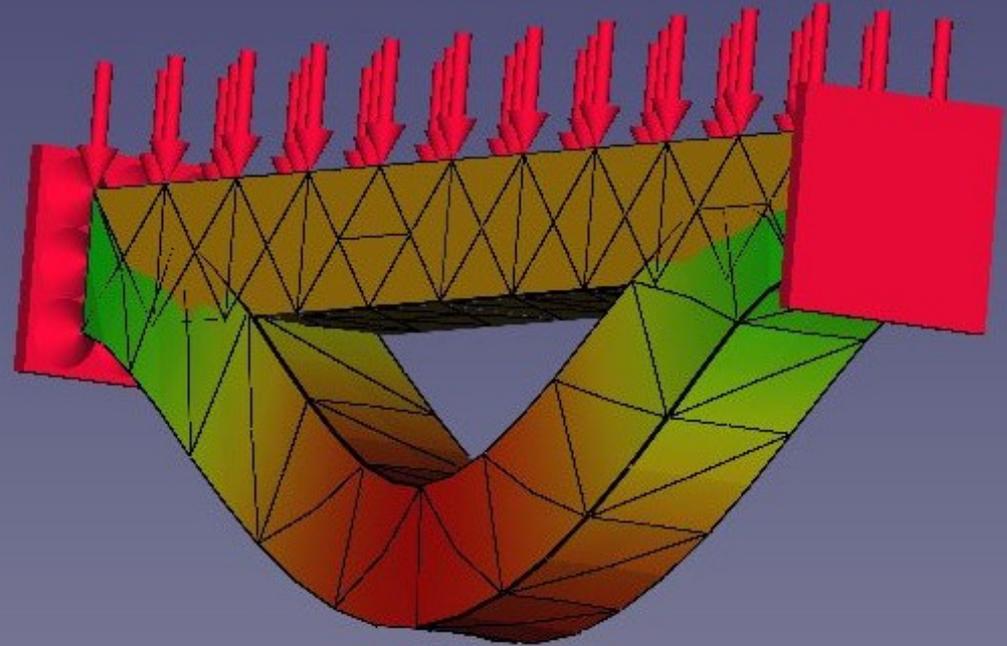
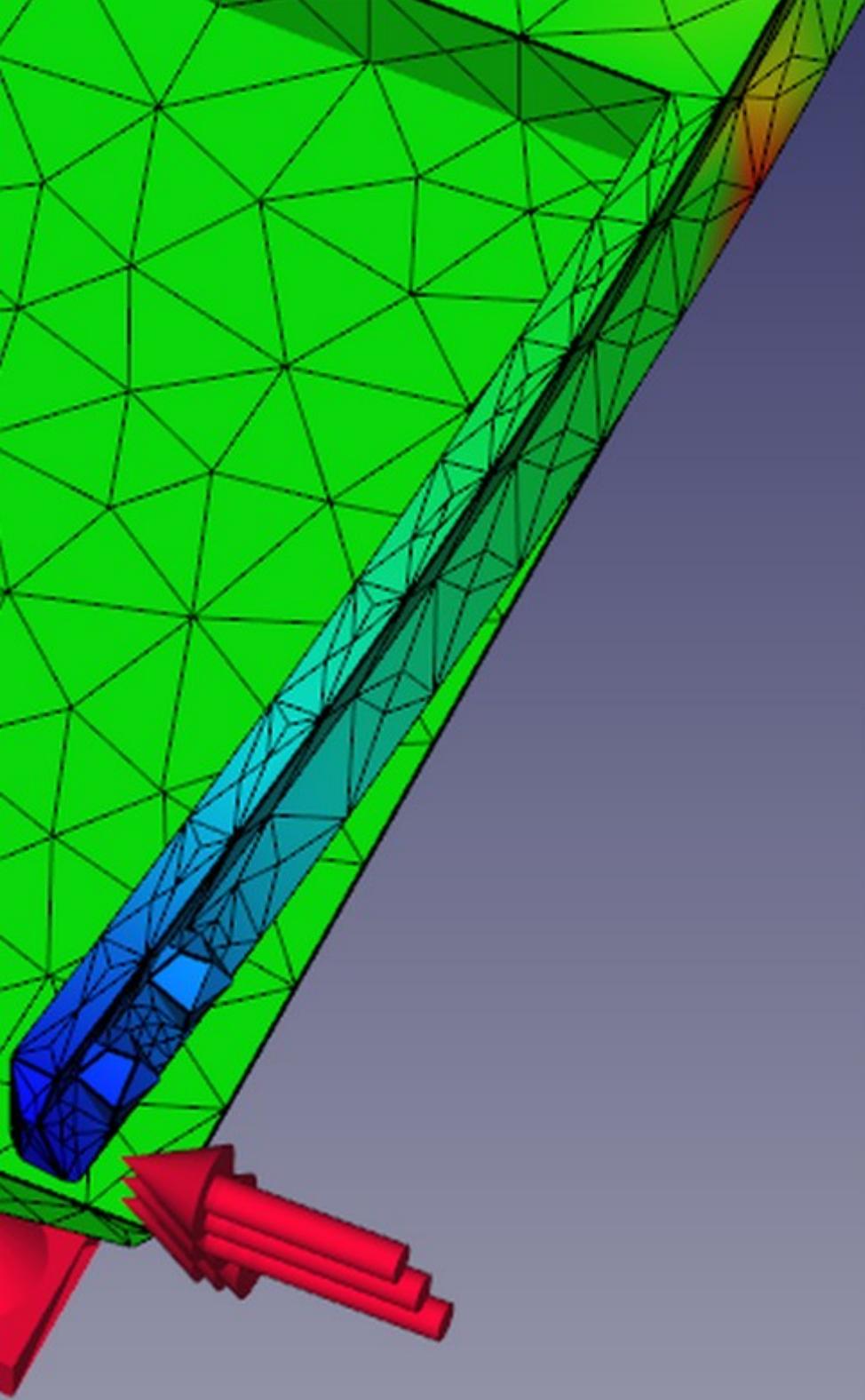
Impressão 2D

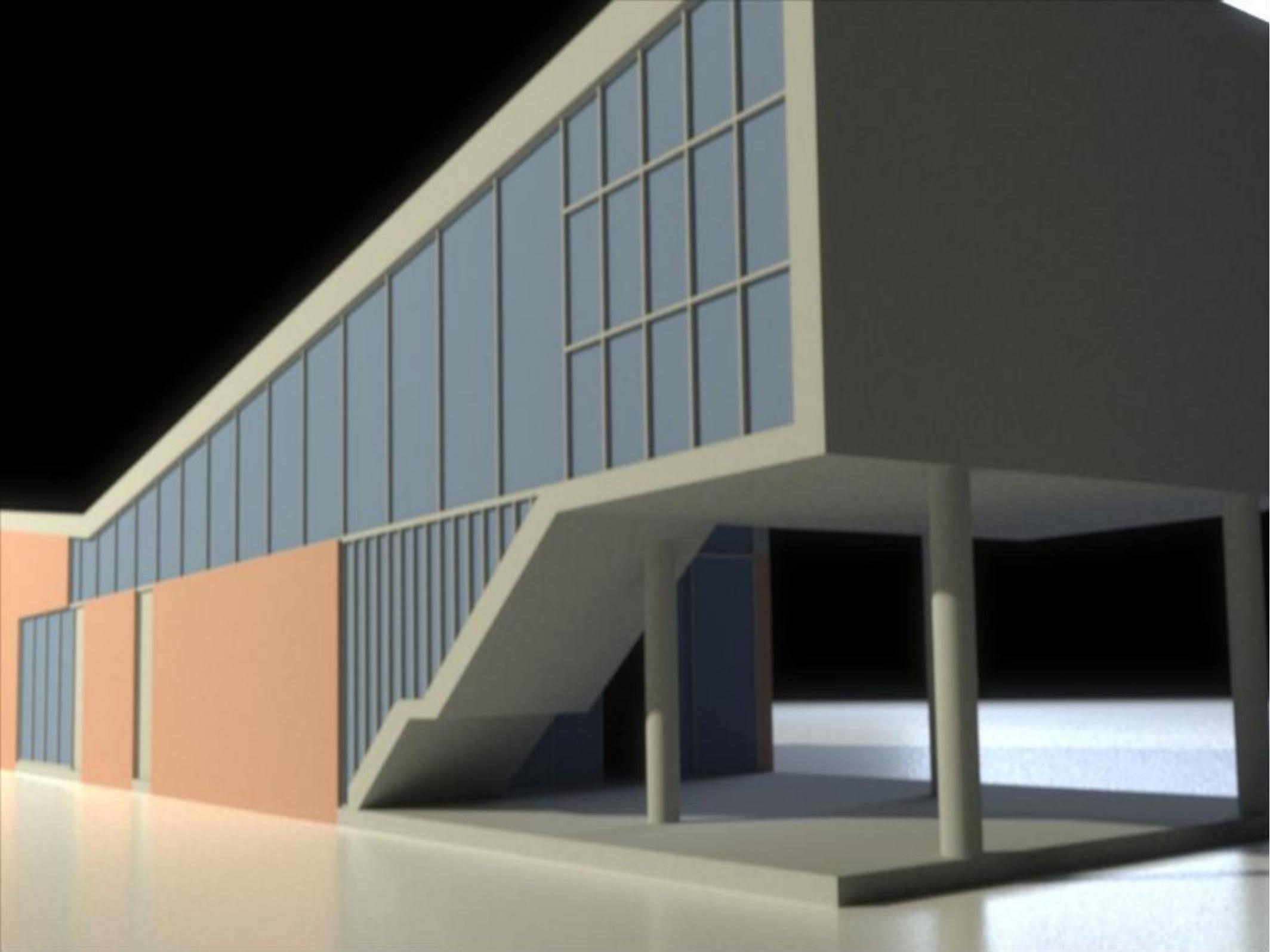
- Drawing: coloca vistas 2D dos objetos em uma página
- Drawing Dimensioning: adiciona cotas, anotações, símbolos, etc
- Exporta para SVG e DXF/DWG



Outros

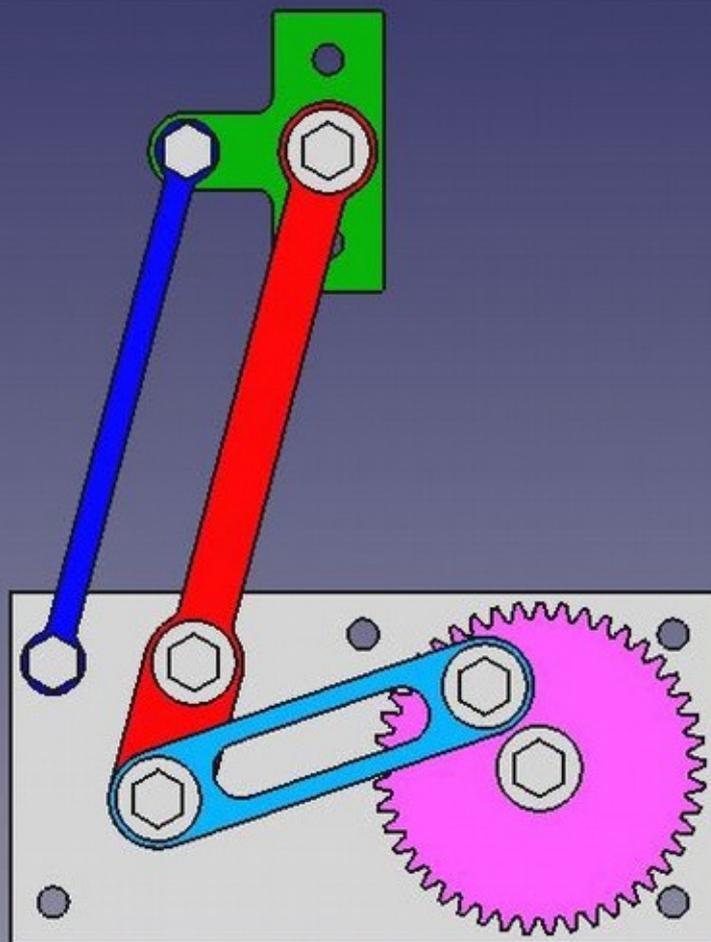
- FEM: Cálculo de esforços internos e deformações
 - Spreadsheet: Planilhas
- Raytracing: produção de imagens realistas





Para já

- Assembly: vínculos e restrições entre objetos diferentes (já temos o Assembly2!)
- Path: Produção de caminhos GCode para máquinas CNC (já funciona!)
- Lista de addons crescendo





Combo View  

Model **Tasks**

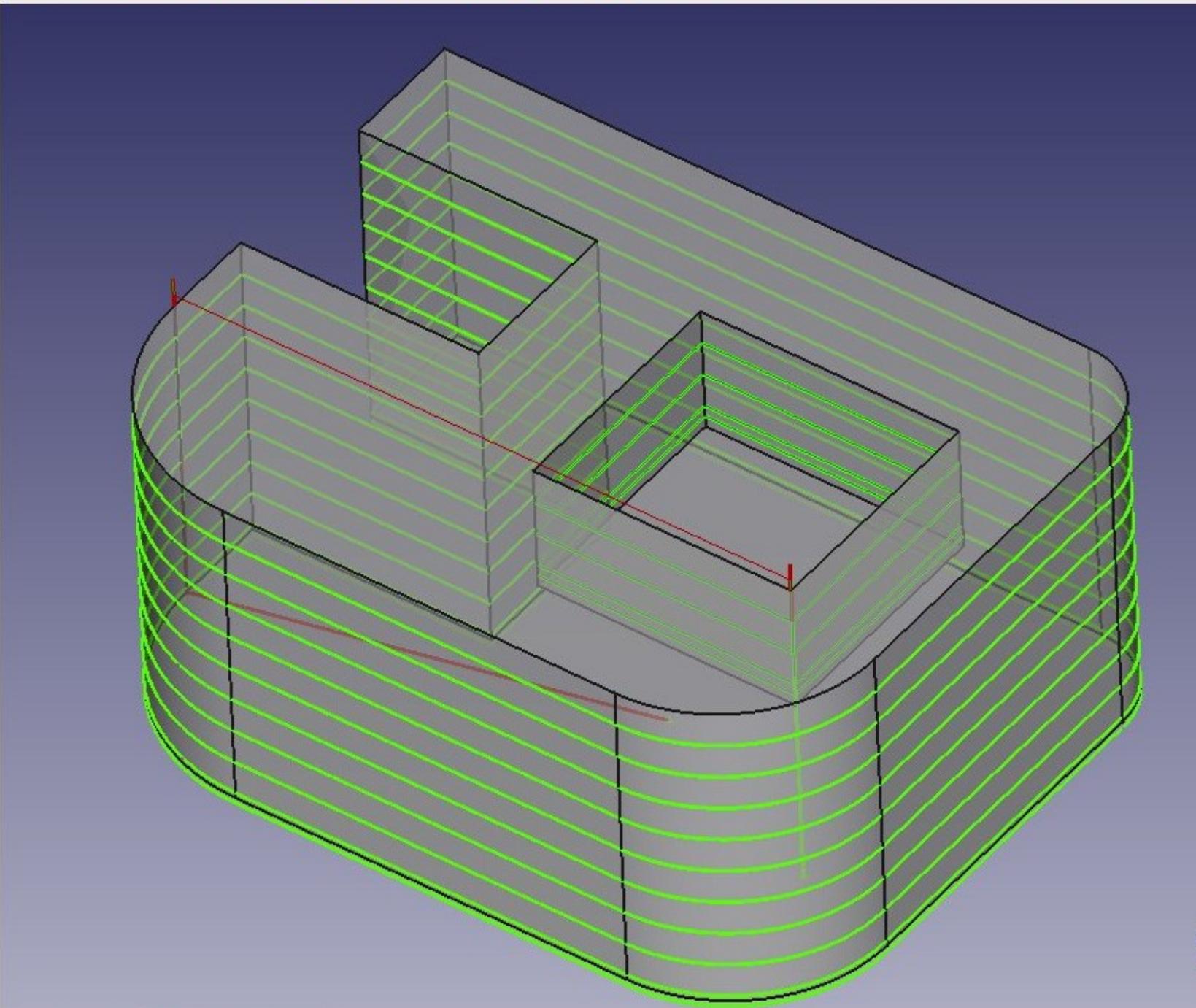
Labels & Attributes

Application

- ▼  **pathtest**
- ▶  Pad
- ▶  Pocket
- ▼  Project
 -  Machine_
 -  Profile
 -  Profile001

Property	Value
----------	-------

Property	Value
----------	-------



View  **Data** 

Start page   pathtest : 1 

Python

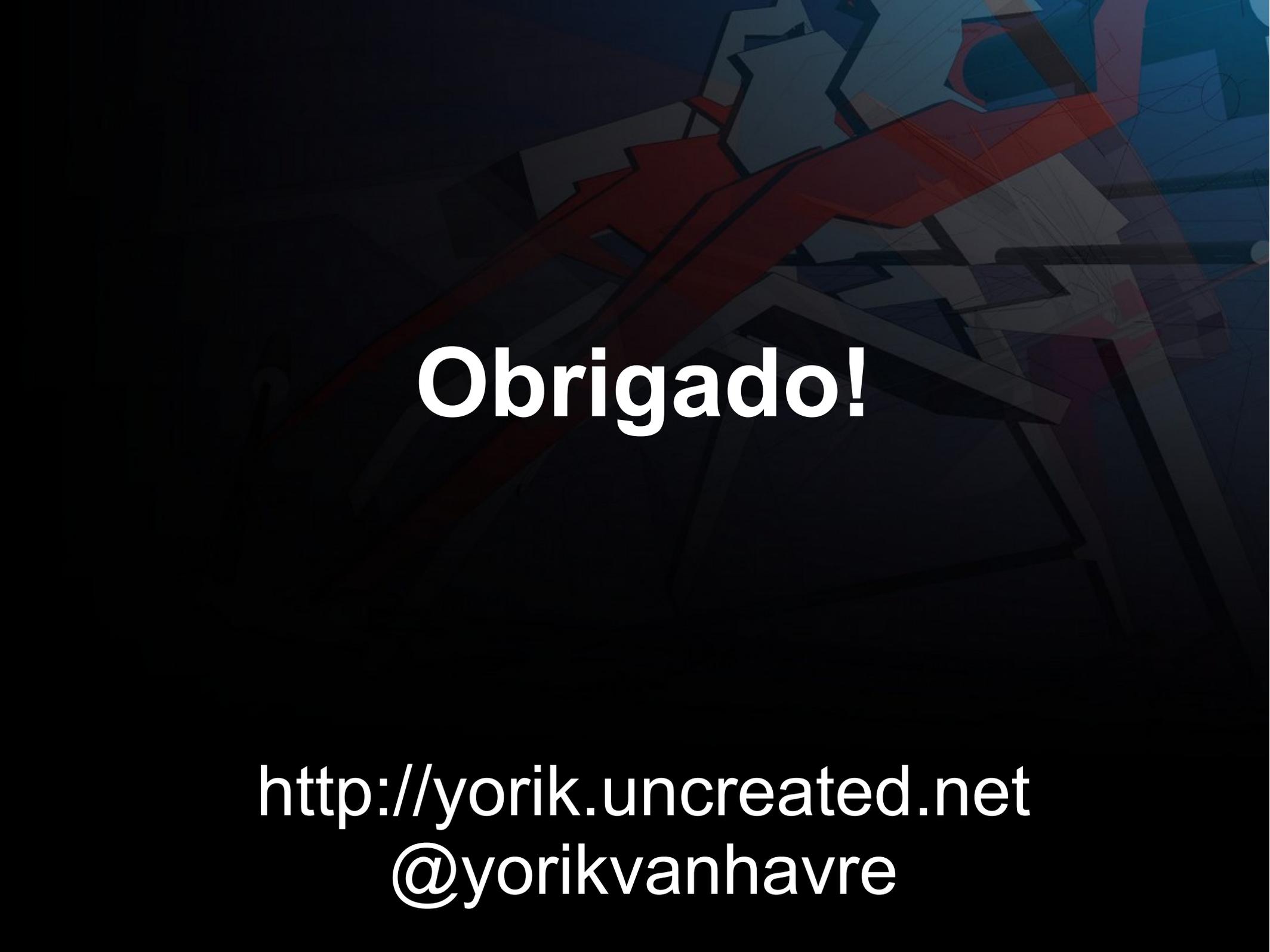
- O FreeCAD é programado em C++, mas o python é usado em todo lugar
 - Python é super fácil de aprender
 - FreeCAD mostra o tempo todo o código python sendo executado (a gente acaba aprendendo sem querer)

<http://www.freecadweb.org>

- Homepage com downloads, etc
 - Documentação (wiki)
- Forum MUITO legal (← vá lá!)
 - Bug tracker

Participe!

- Precisamos de gente na seção em português do forum!
- Ajude a traduzir a documentação (ótima maneira de aprender)
- Use, recomende, publique dicas, vídeos, tutoriais, etc.



Obrigado!

<http://yorik.uncreated.net>
[@yorikvanhavre](#)