Freecad

Introduktion

- Pågående utveckling, senaste v0.17
- Rotera - mitten och vänster musknapp
- Workbenches styr vilka knappar som finns tillgängliga

Vad kan Freecad

- Importera/exportera stl,step,iges...
- Skapa parametriserade geometrier
- Mäta
- Enklare FEM mha Calculix (utvecklats på senare tid)
- Python gränssnitt

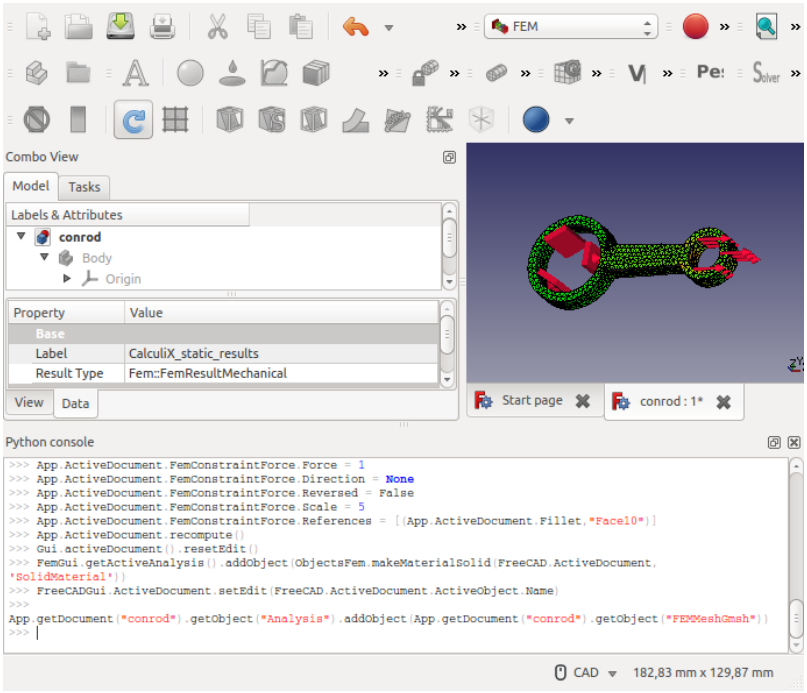

Figure 1: *python*

- Batch-körningar

  $freecadcmd hello.py

  hello!

Vad kan inte Freecad

- Importera Catia-filer

- Assemblera (utvecklingen verkar ha somnat här)
- Inte så bra på långa beroendekedjor

Filformat

- IGES - funkar sådär
- STEP - (ISO 10303-21) Riktigt bra
- .catpart etc. inte alls!
- .stl - lista med facetter. Inga features

Övning

Part Design

- Skapa sketch - Välj Workbench Part Design. Klicka New Sketch

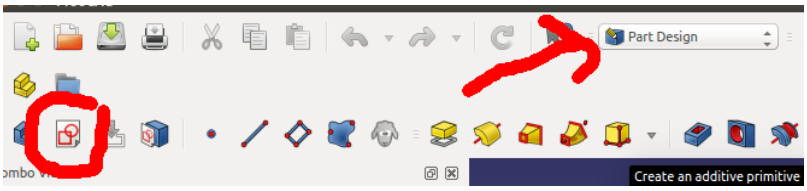

Figure 2: *Skapa sketch*

- Rita fyra cirklar med linjer emellan

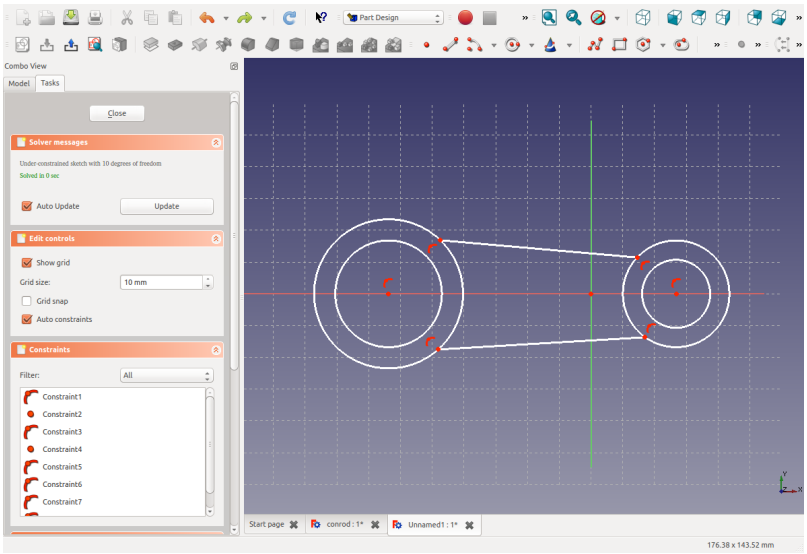

Figure 3: *cirklar med linjer*

- Trimma de yttre cirklarna

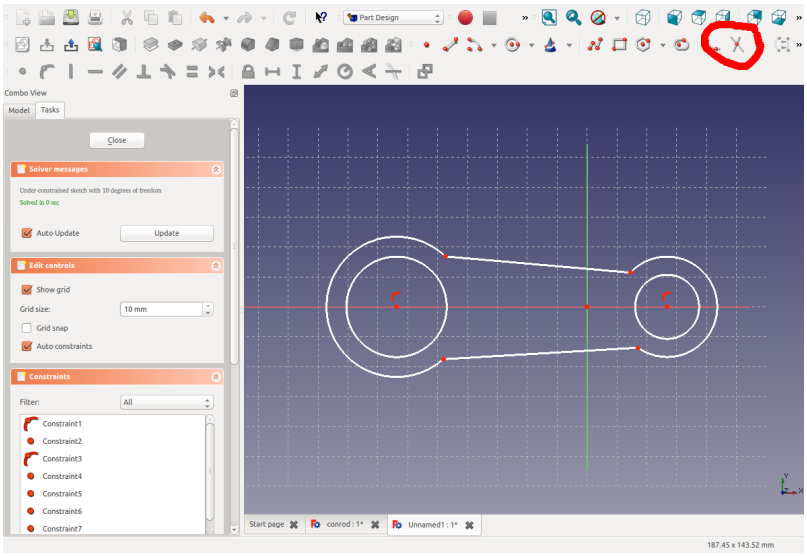

Figure 4: *trimma*

- Inför "hjälplinjer"

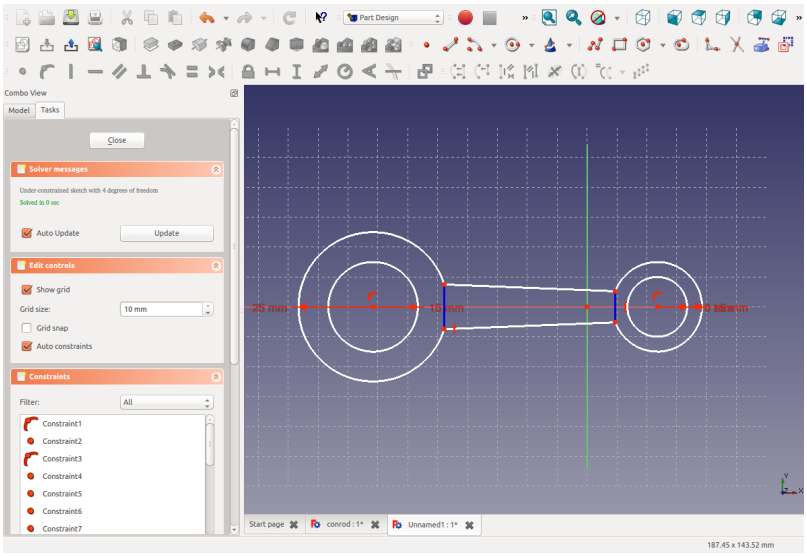

Figure 5: *hjälplinjer*

- Sätt constraints
- Extrudera (Pad)

Part

- Fillet

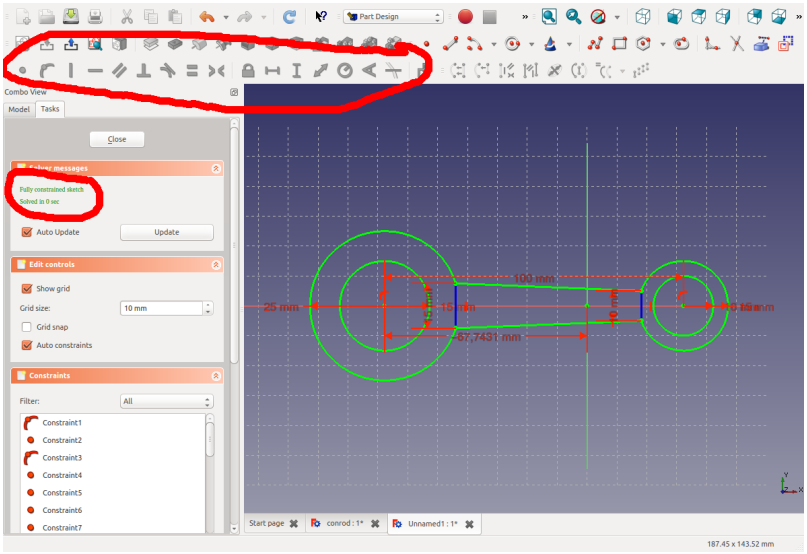

Figure 6: *constraints*

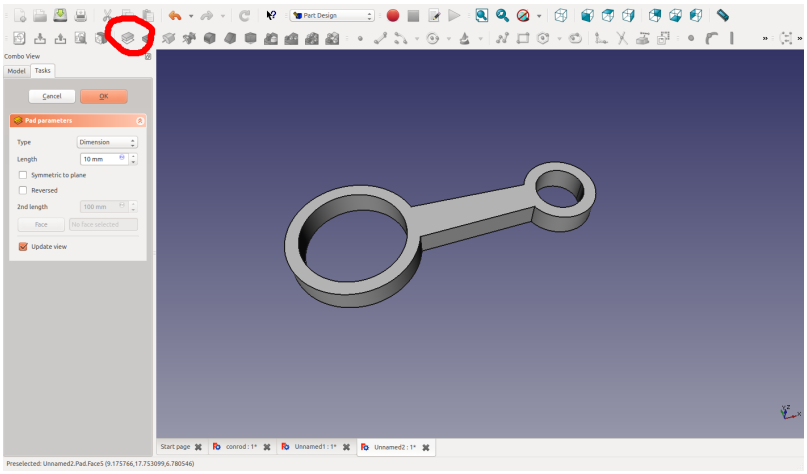

Figure 7: *Extrudera*

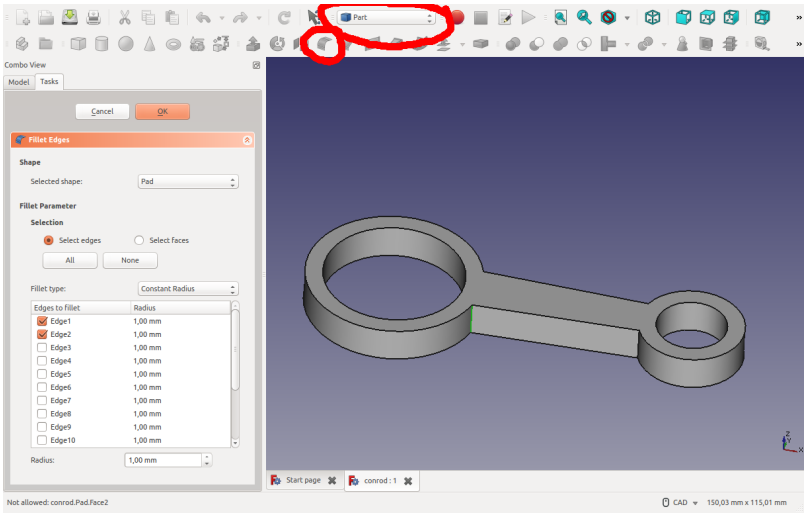

Figure 8: *fillet*

FEM

- Ny mesh

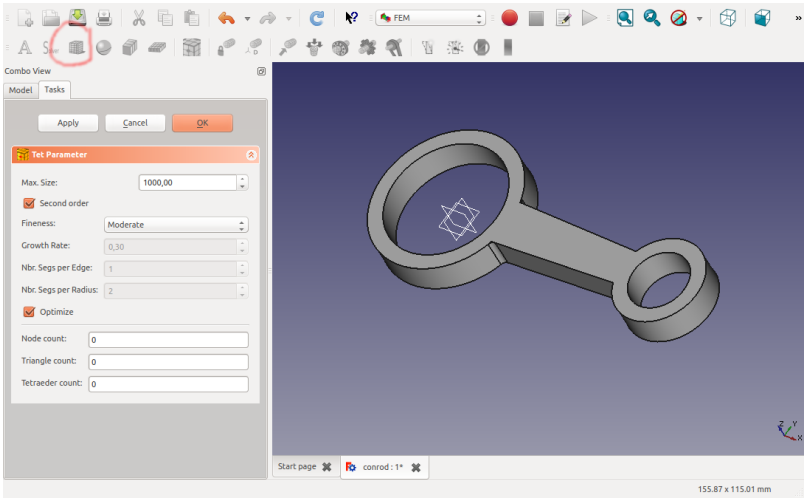

Figure 9: *Ny mesh*

- Ny Analys
- Material
- Create FEM constraint

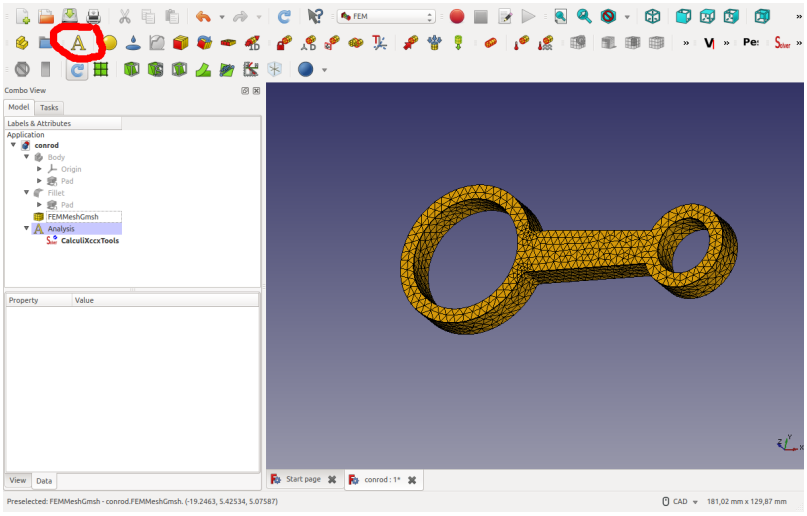

Figure 10: *Ny Analys*

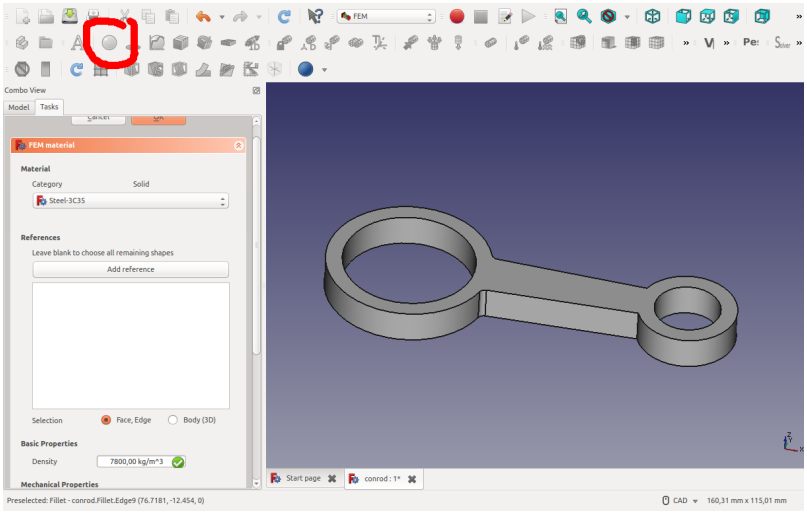

Figure 11: *material*

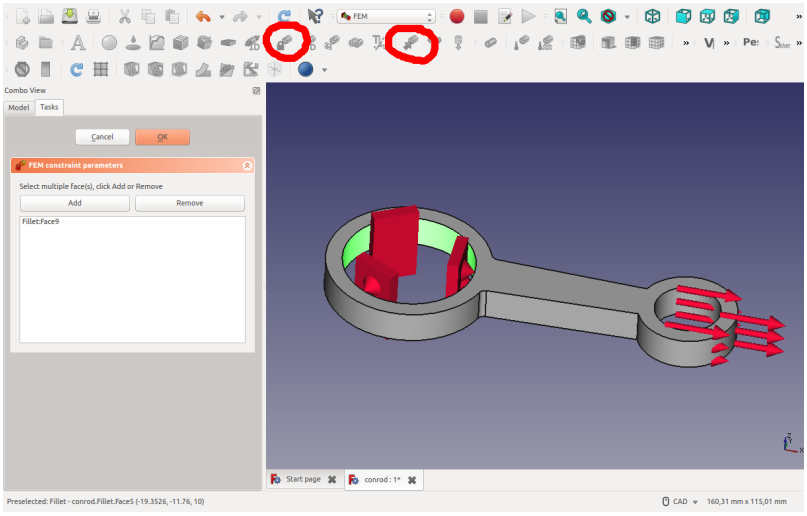

Figure 12: *bc*

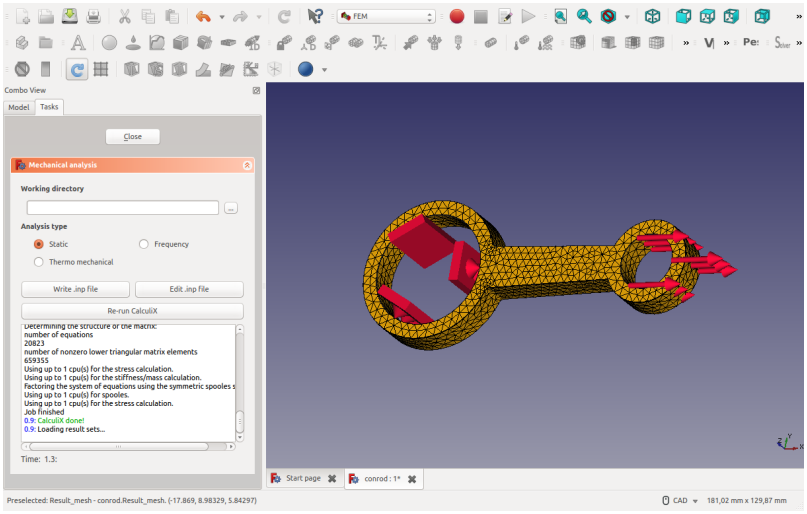

Figure 13: *Lös*

- Lös (flytta meshen till analysen)

- Post

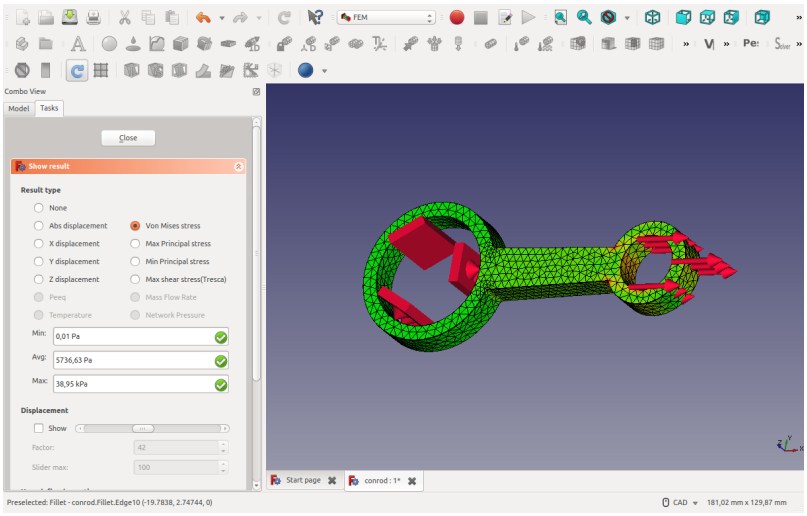

Figure 14: *post*

Python

- Skriv direkt i “Python Console”
- Exekvera en fil

>>> execfile(“your_path_to_script”)

- Kör utan GUI

  $freecadcmd filename.py

Python Exempel

Skapat en svets från uppmätt svetsgeometri

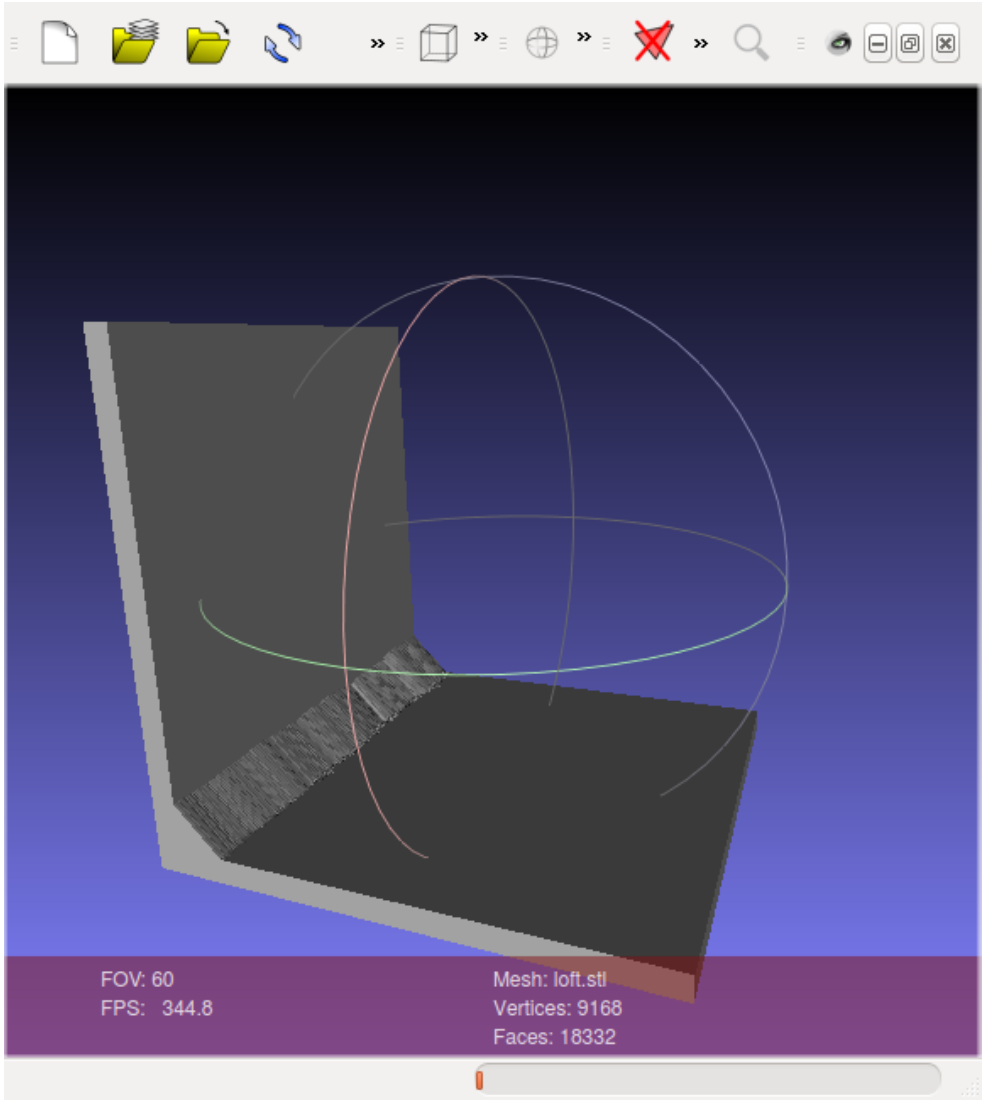

Figure 15: *Svets*

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