

Desenho e Tecnologia Mecânica

LOM 3093

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AULA

Execução de desenho com sistema CAD.

REVISÃO

ENGENHARIA REVERSA

O que está já está pronto, e descrevo o projeto (volto o processo de produção).

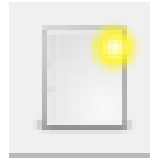
Todo desenho vem da engenharia reversa.

FreeCAD

Rotina Part Design:

Desenhando um plano:

- Ficheiro / Novo;



- Ver / Workbench / PartDesign;


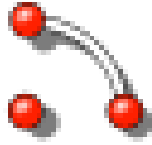



- Acionar função: Criar Esboço.



FreeCAD

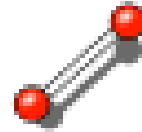
PartDesign Workbench (Geometrias):

- Ponto 
- Arco 
- Círculo 

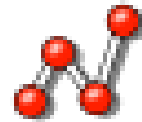
FreeCAD

PartDesign Workbench (Geometrias):

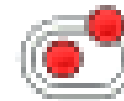
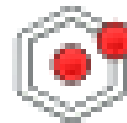
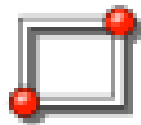
- Linha – 2 pontos



- Várias linhas






- Retângulo, sextavado ou obilongo



FreeCAD

PartDesign Workbench (Regras e definições):

- Restrição 
- Coincidência 
- Ponto num objeto 

FreeCAD

PartDesign Workbench (Regras e definições):

- Distância horizontal / vertical



- Vertical / Horizontal



- Comprimento





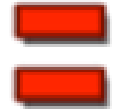

FreeCAD

PartDesign Workbench (Regras e definições):

- Raio 
- Paralelo 
- Perpendicular 

FreeCAD

PartDesign Workbench (Regras e definições):

- Ângulo interno 
- Tangente 
- Comprimento equivalente 
- Simetria 

FreeCAD

PartDesign Workbench (Outros):

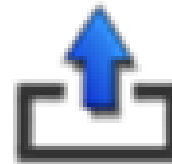
- Novo sketch



- Visualizar sketch



- Sair do sketch



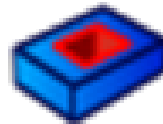
FreeCAD

PartDesign Workbench (Ferramentas de construção):

- Encher



- Retirar



- Revolucionar

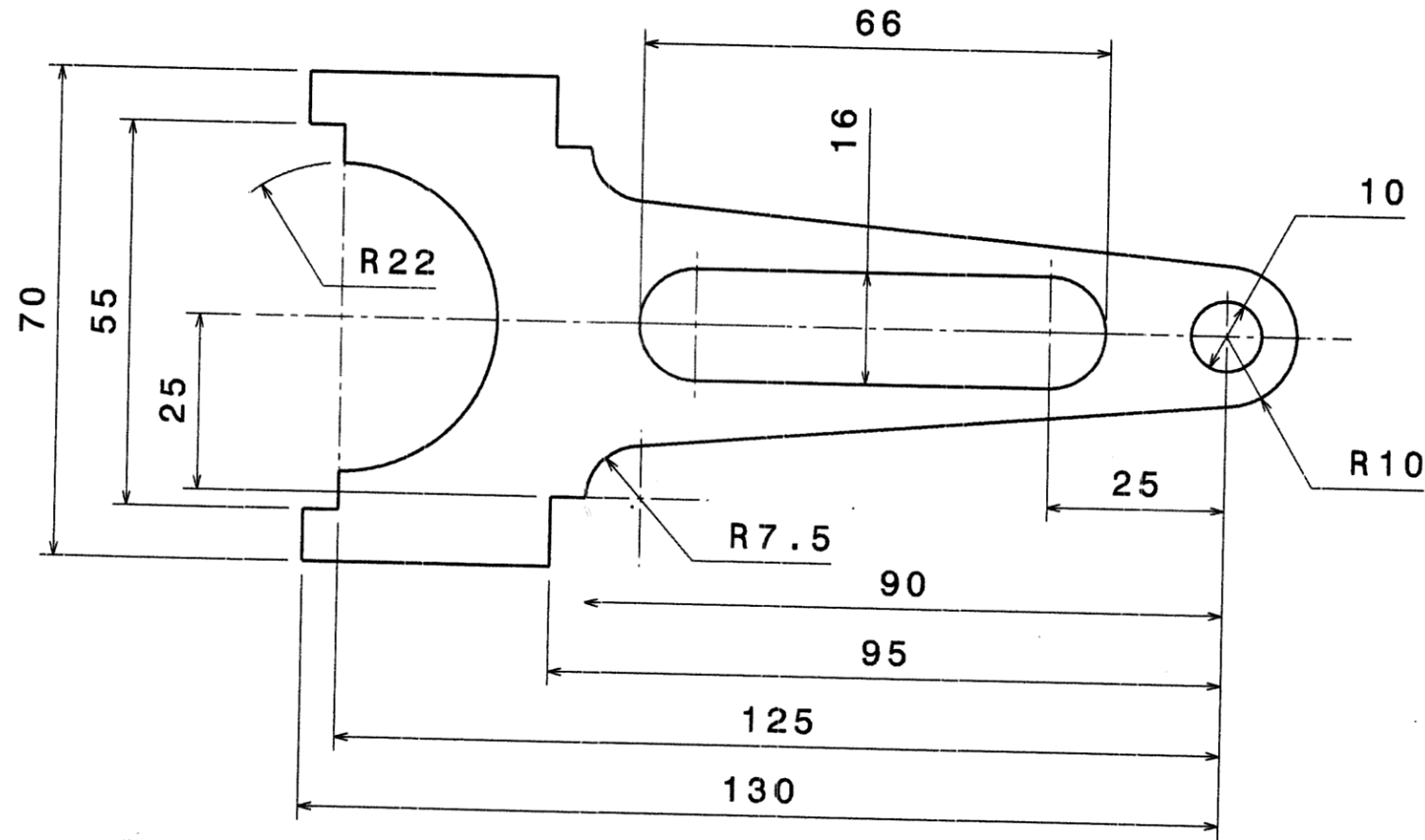


- Entralhar



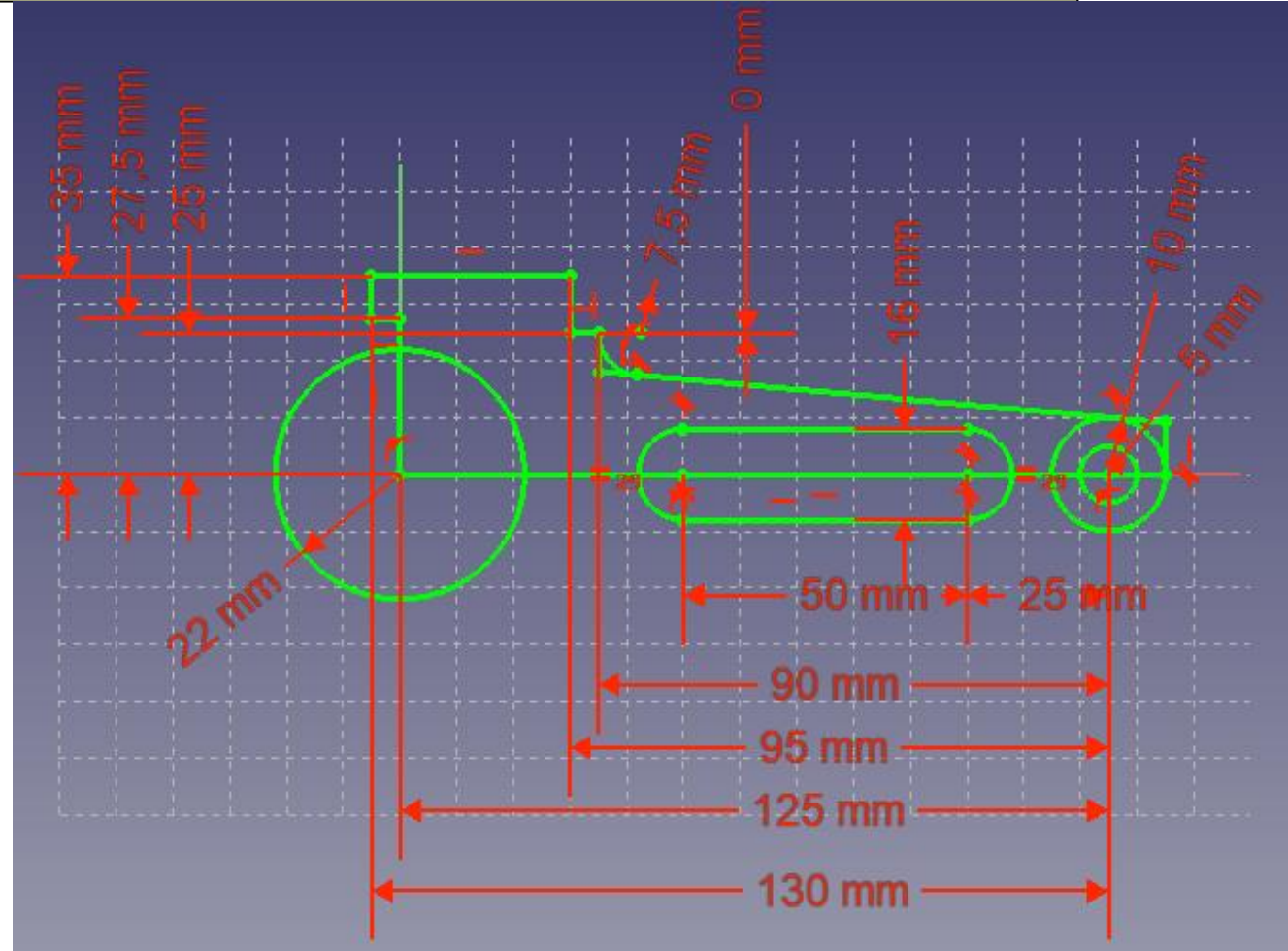
FreeCAD

Criar a seguinte figura:



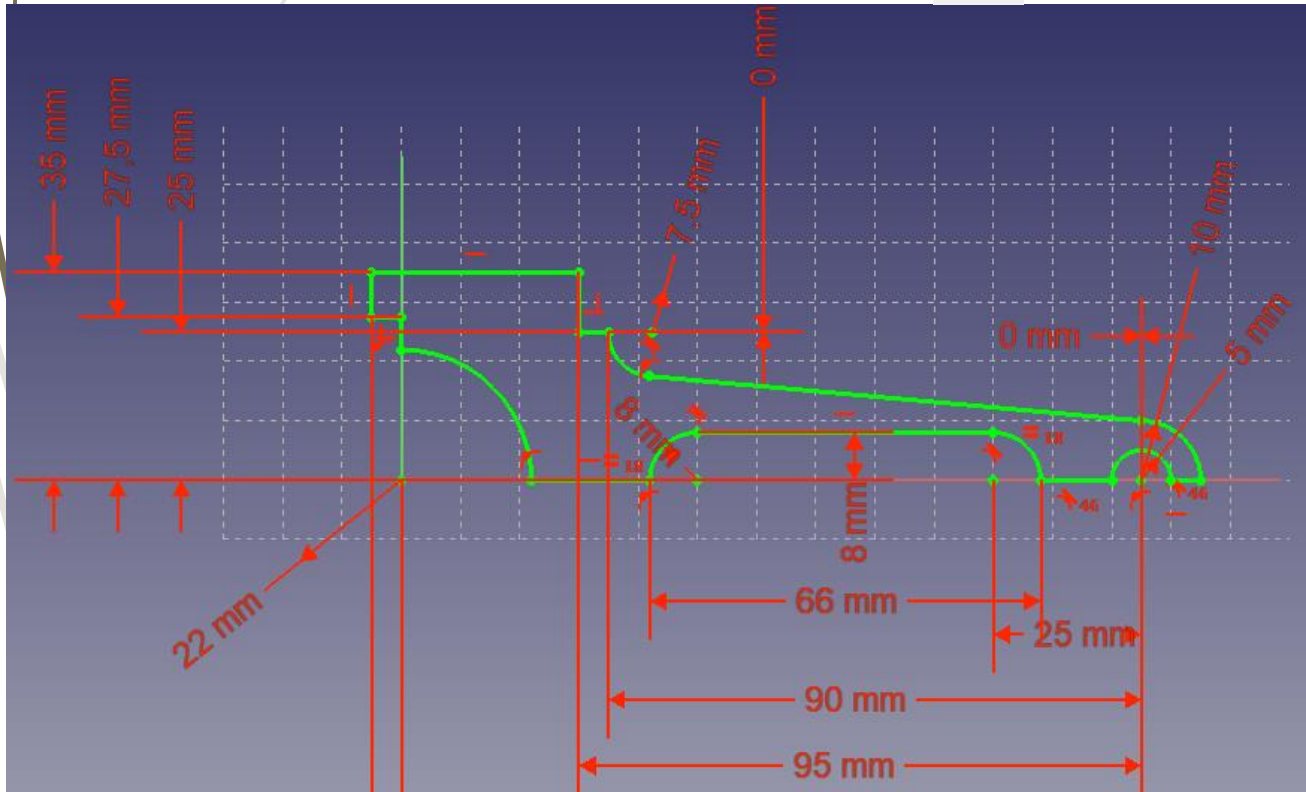
PROCEDIMENTO

Etapa 2 – Colocar suas regras e definições.



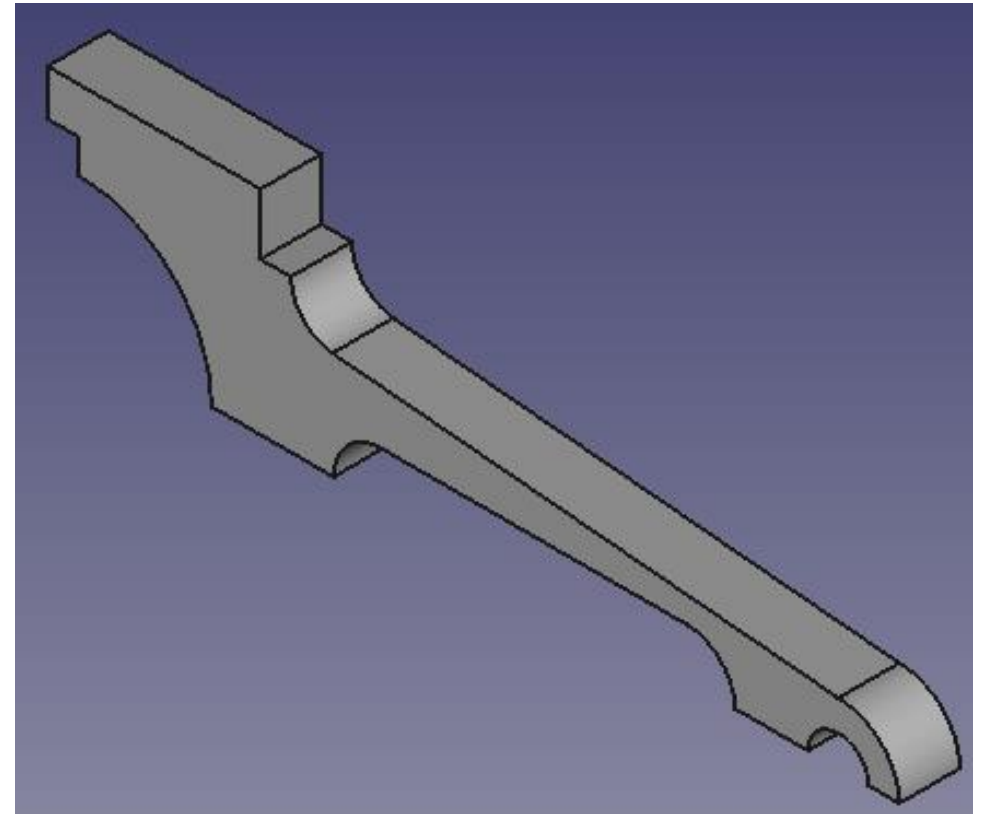
PROCEDIMENTO

Etapa 3 – Realizar o corte para definir perfil (recortar – trim ).



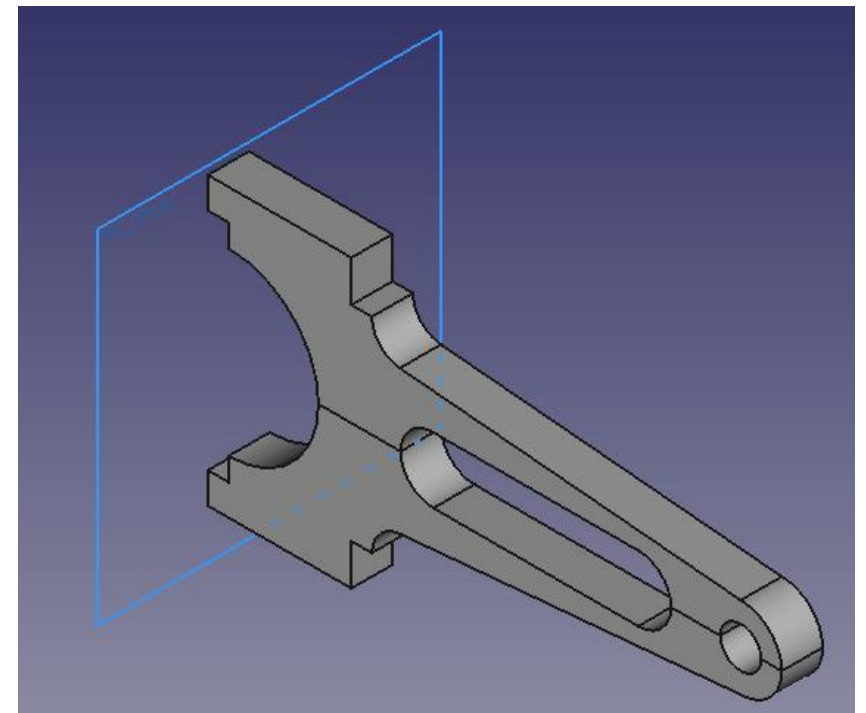
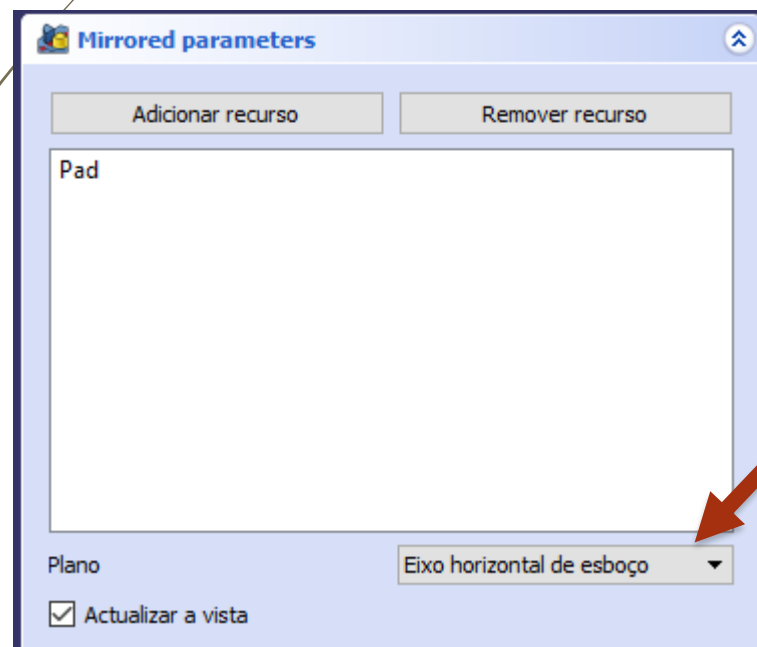
PROCEDIMENTO

Etapa 4 – Criar o sólido.



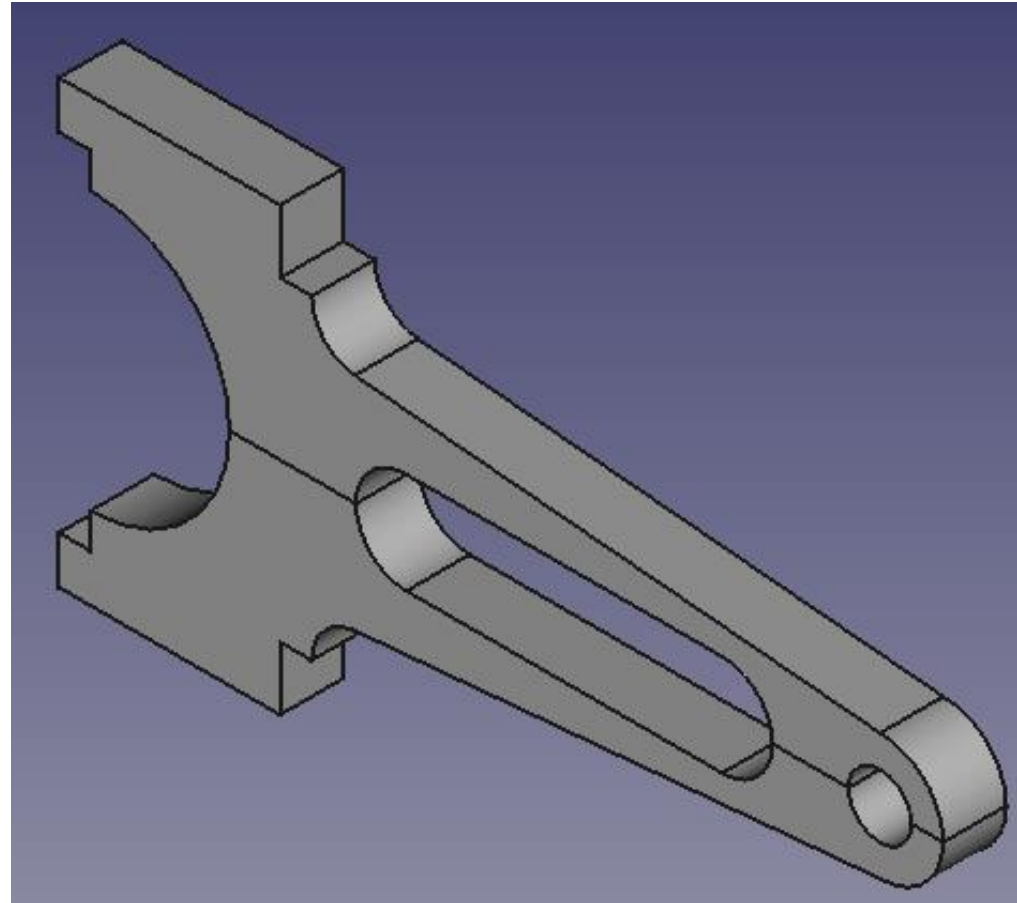
PROCEDIMENTO

Etapa 5 – Aplicar a função espelho (mirror  - horizontal) no sólido.



PROCEDIMENTO

PROJETO FINAL



FreeCAD

Bom trabalho