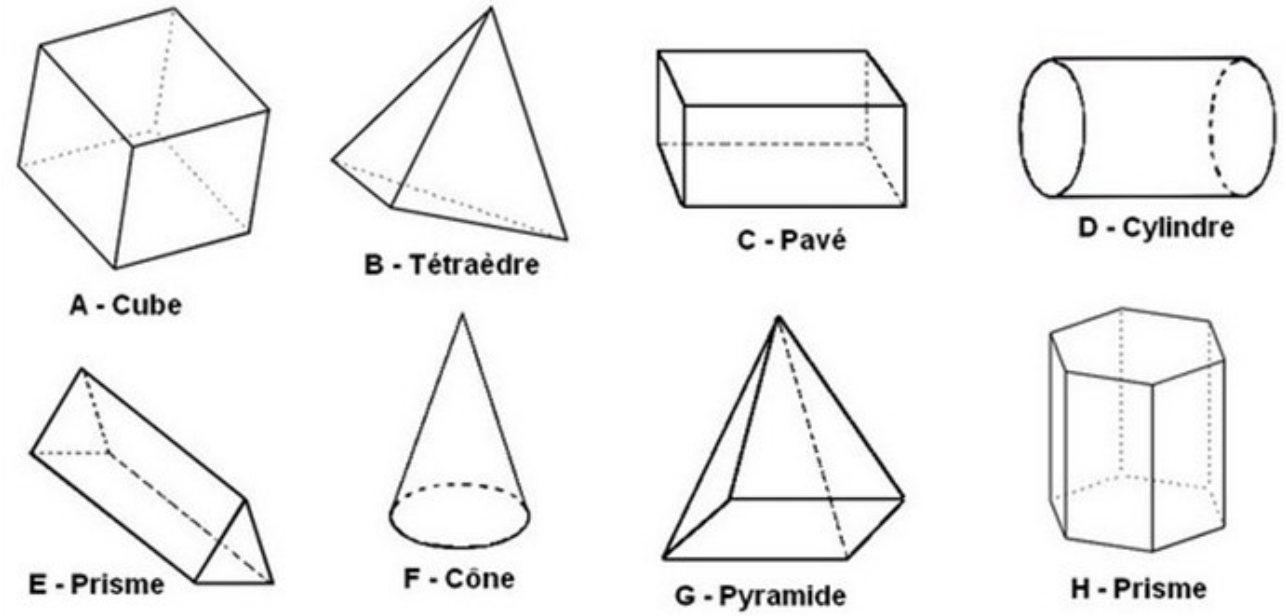
ACTIVITE SUPPLEMENTAIRE SOLIDWORKS

"activité 1 - formes géomètriques sur solidworks"

Ci-dessous les formes géométriques simples qu'il va falloir représenter sur **solidworks**.



Objectifs :

- modéliser sur **solidworks** les formes géométriques.
- il faut respecter le cahier des charges de chacun des volumes élèmentaires.
- donner la formule du volume de chacune des formes.

Exercice 1 : formules des volumes

V (cube-A) =

V (tétraèdre-B) =

V (pavé-C) =

V (cylindre-D) =

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V (prisme-E) =

V (cône-F) =

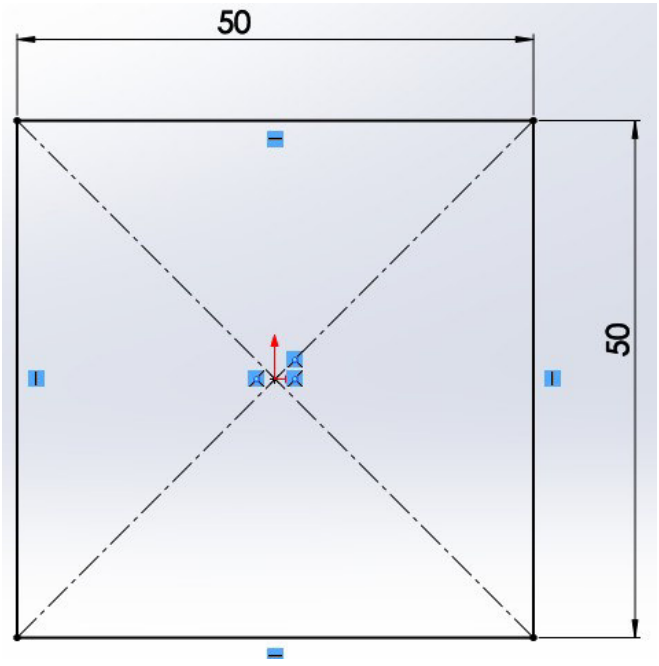
V (pyramide-G) =

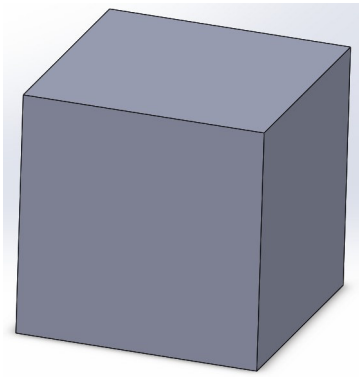
V (prisme-H) =

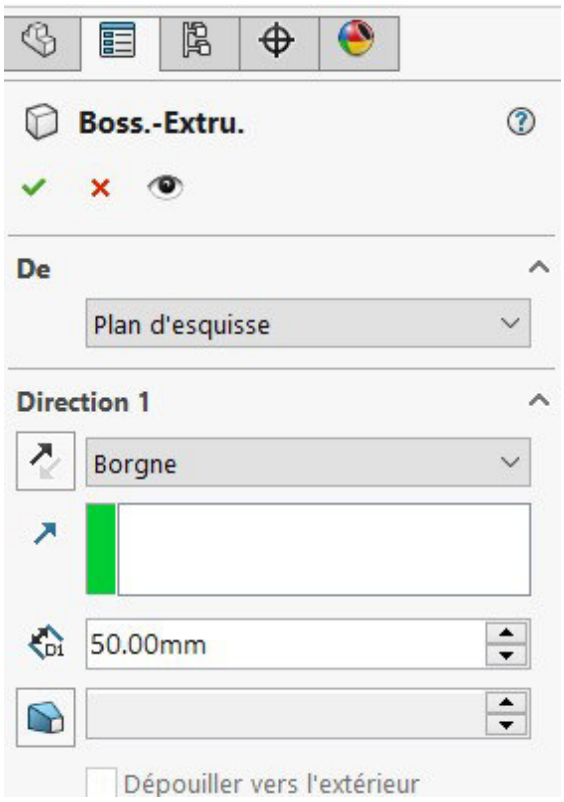
Exercice 2 : modéliser les formes géométrique sur solidworks

consigne : reproduire les formes géométriques en respectant les dimensions.

Dimensions CUBE – A : 50 mm x 50 mm x 50 mm







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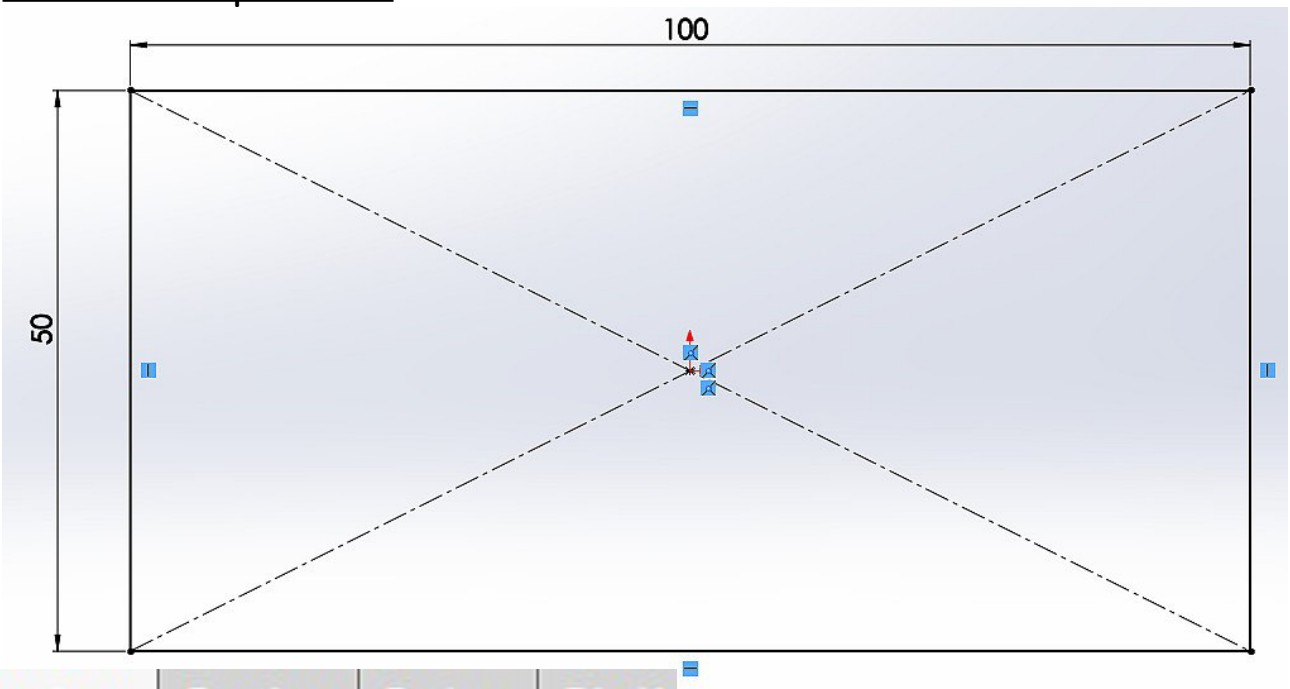
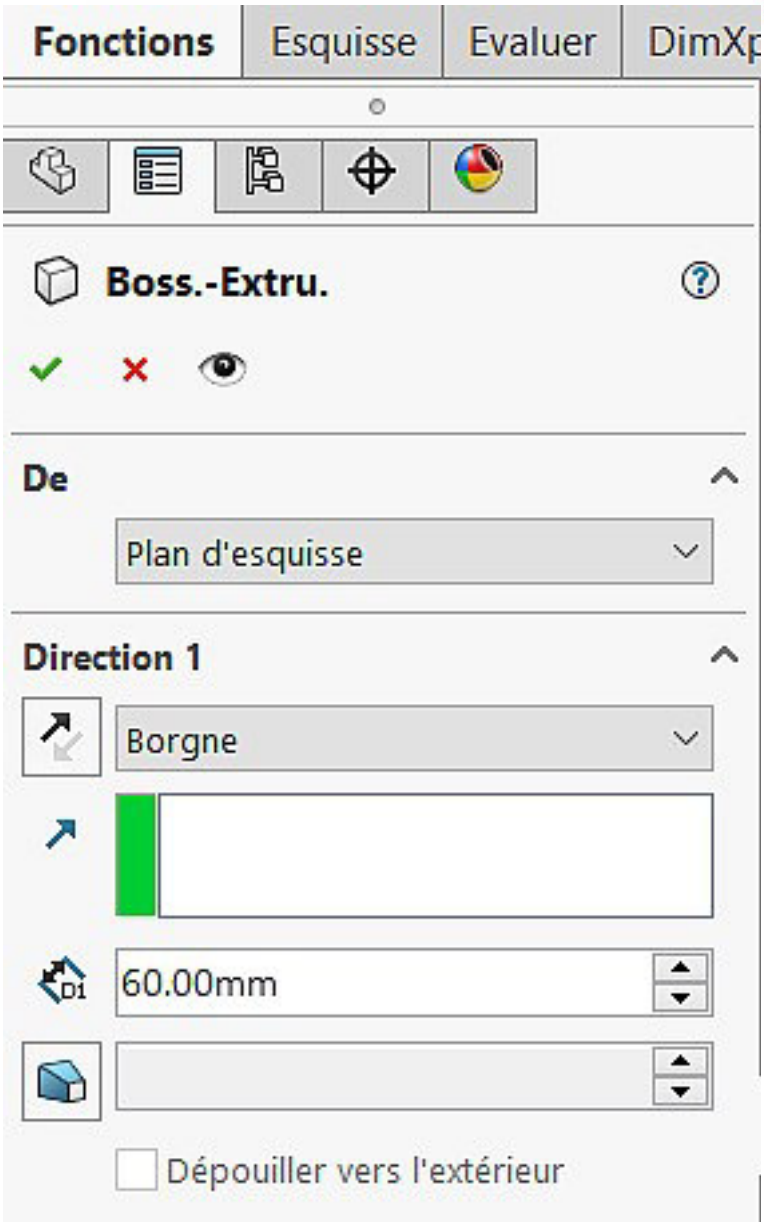
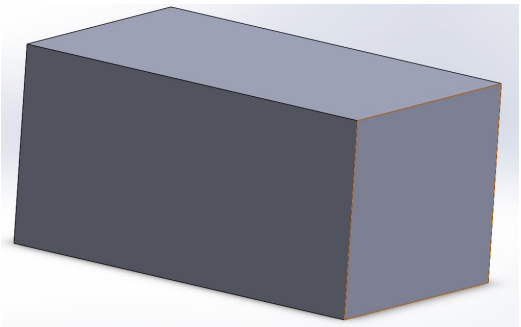
Dimensions pavé- C: 100 mm x 50 mm x 60mm







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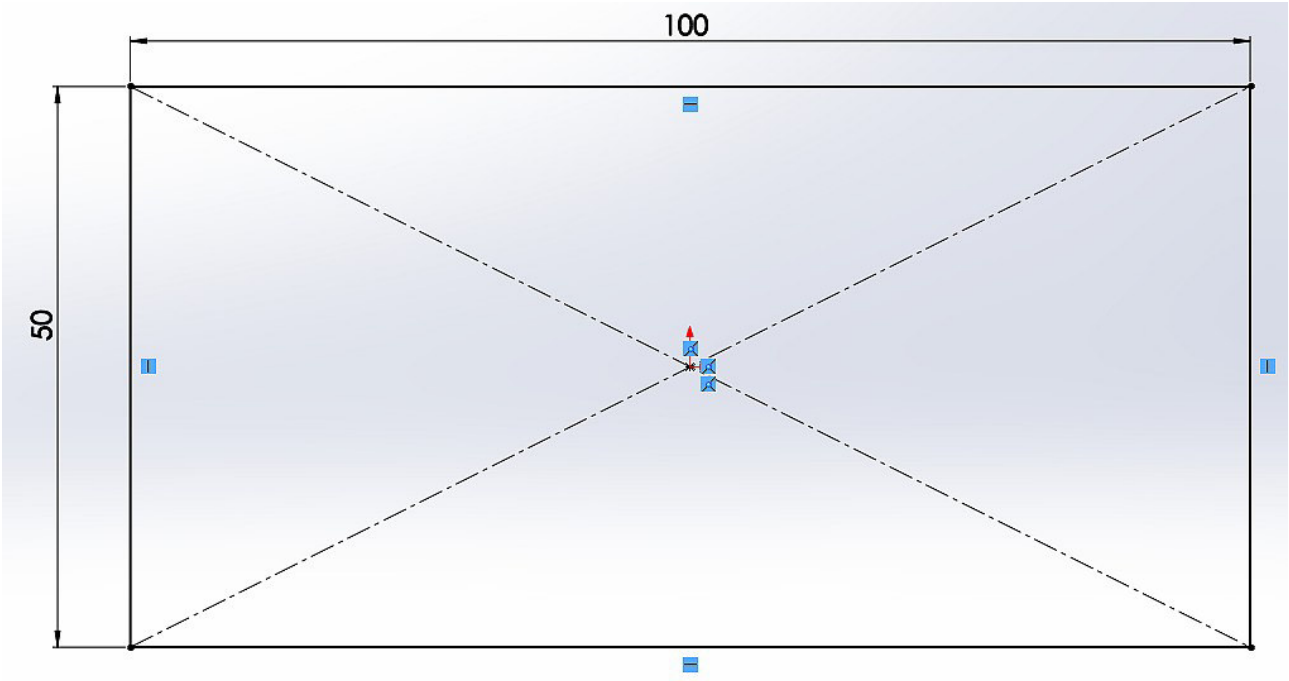
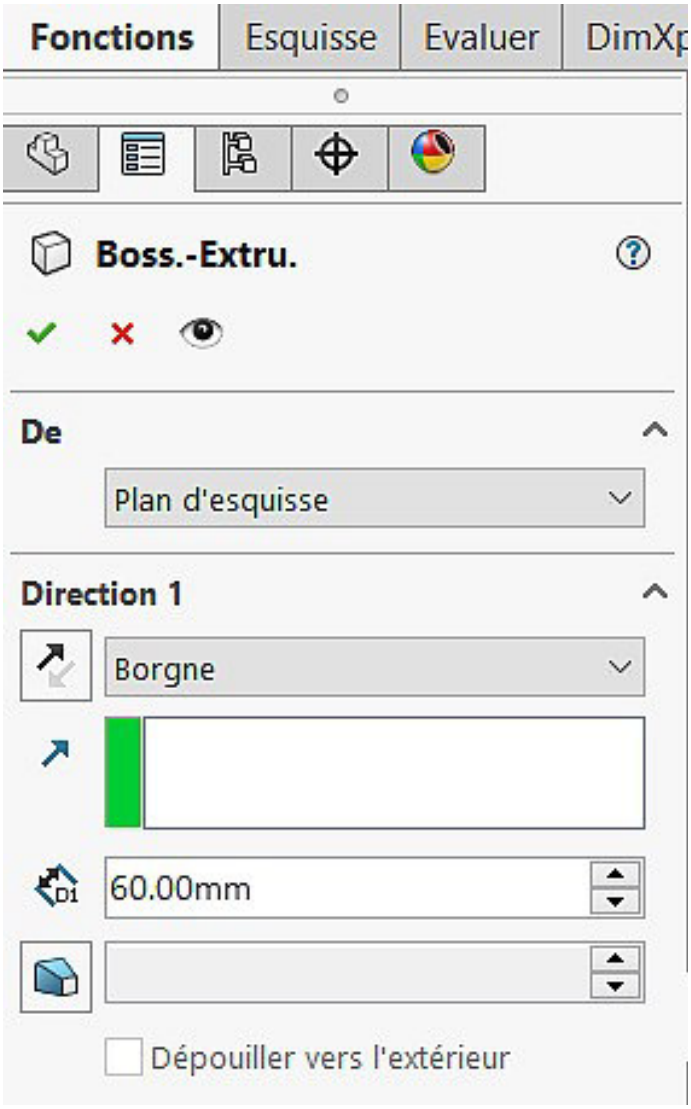
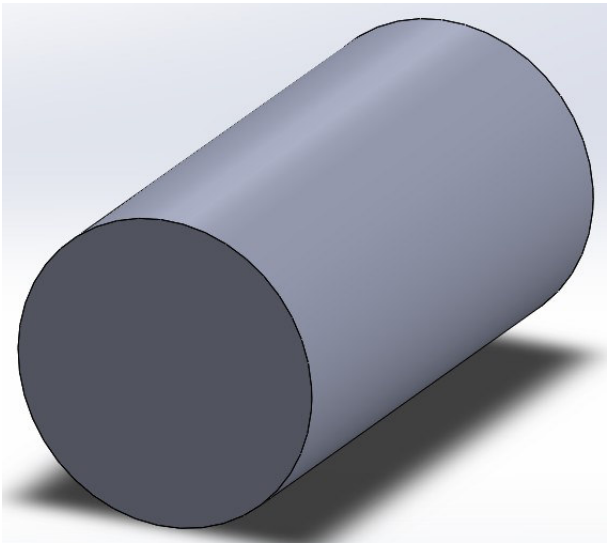
Dimensions cylindre-D : diamètre 50 mm x longueur 100 mm







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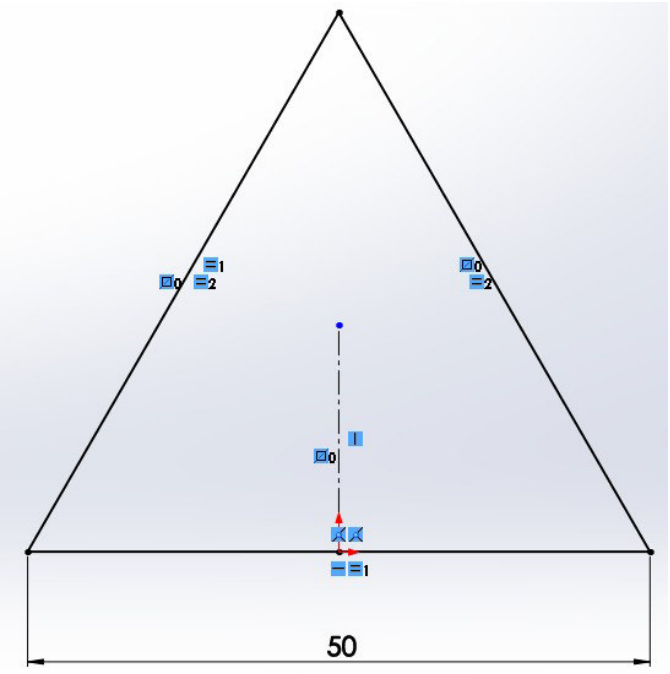
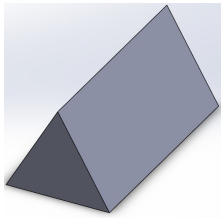
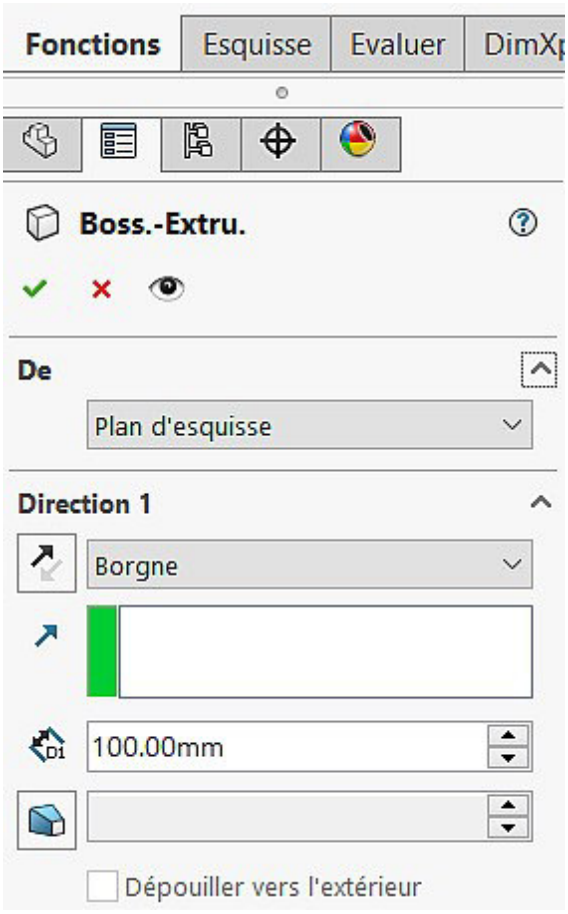
Dimensions prisme-E :

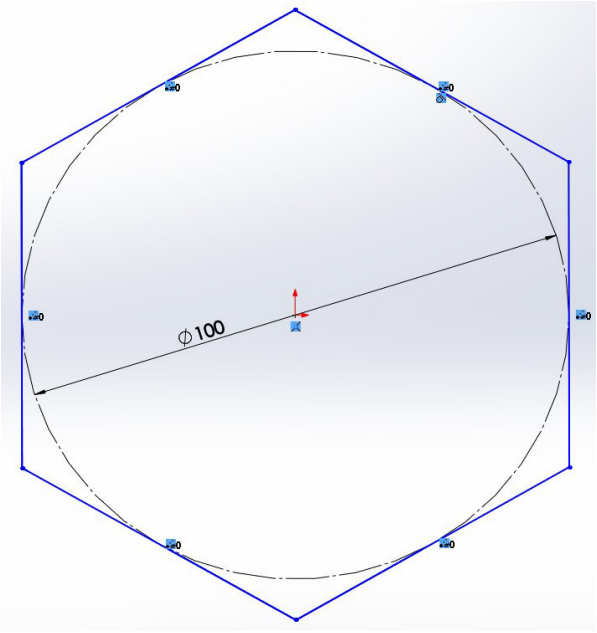
triangle equilatéral de 50 mm -extrusion 100mm

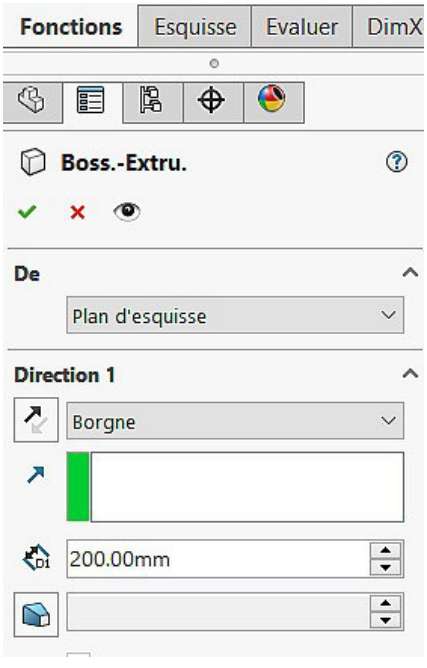


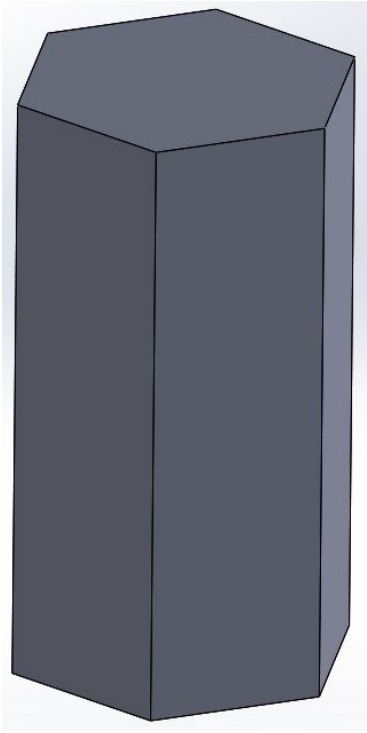




Dimensions prisme-H :







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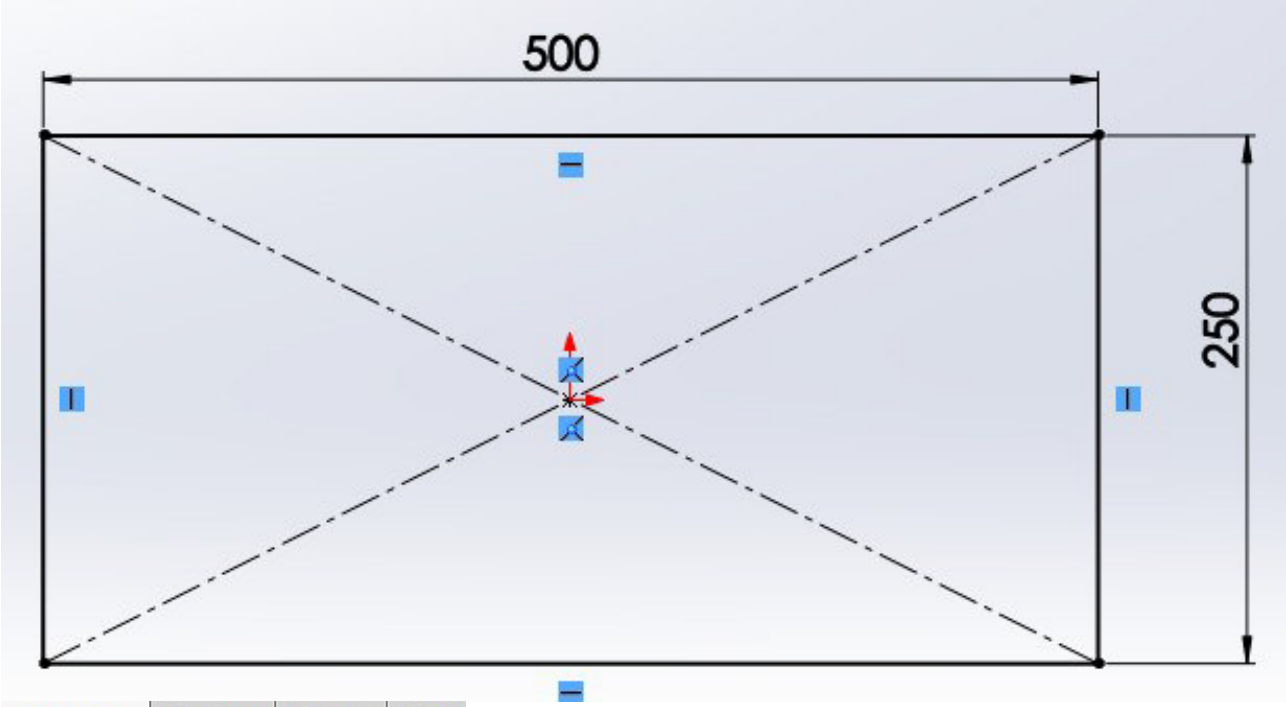
ACTIVITE SUPPLEMENTAIRE SOLIDWORKS

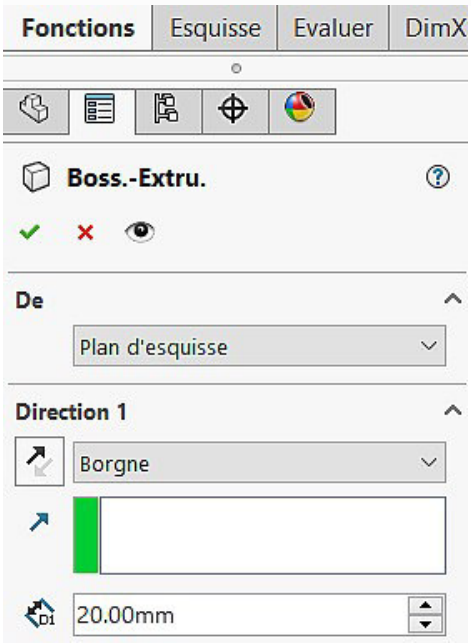
"activité 2 – meuble simple sur solidworks "

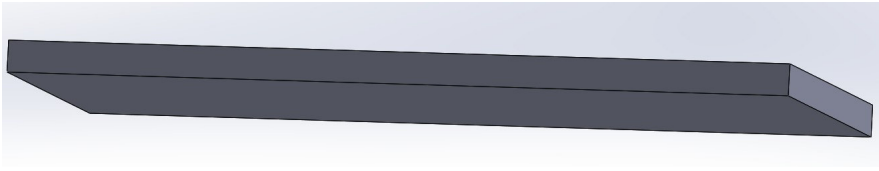
objectifs :

1. suivre les consignes de conception
2. concevoir sur solidworks un meuble simple
3. savoir utiliser les outils esquisse,extrusion

1/ conception d'un meuble simple







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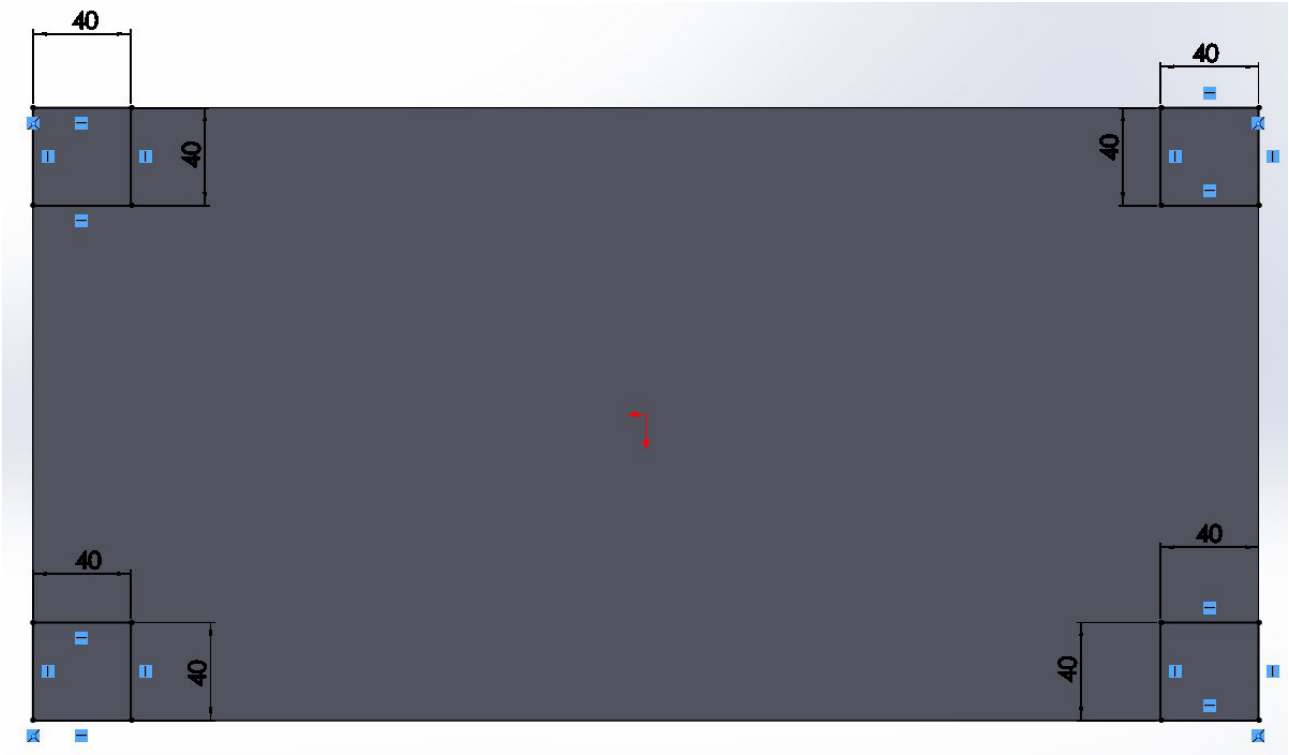
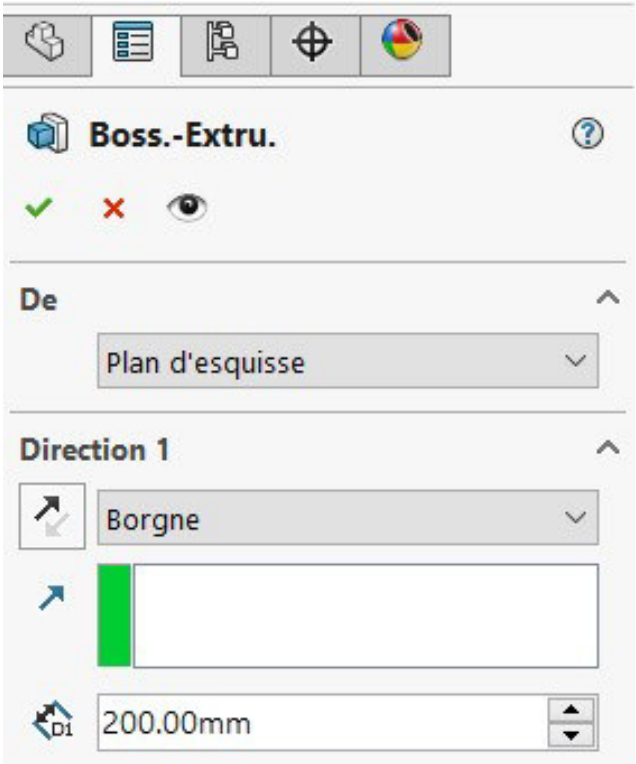
- réaliser une esquisse sur la face supérieure
- représenter 4 carrés de 40 mm x 40 mm



réaliser une extrusion sur 200 mm des 4 pieds de table.



(dimensions non réaliste)

TABLE REALISEE SUR SOLIDWORKS

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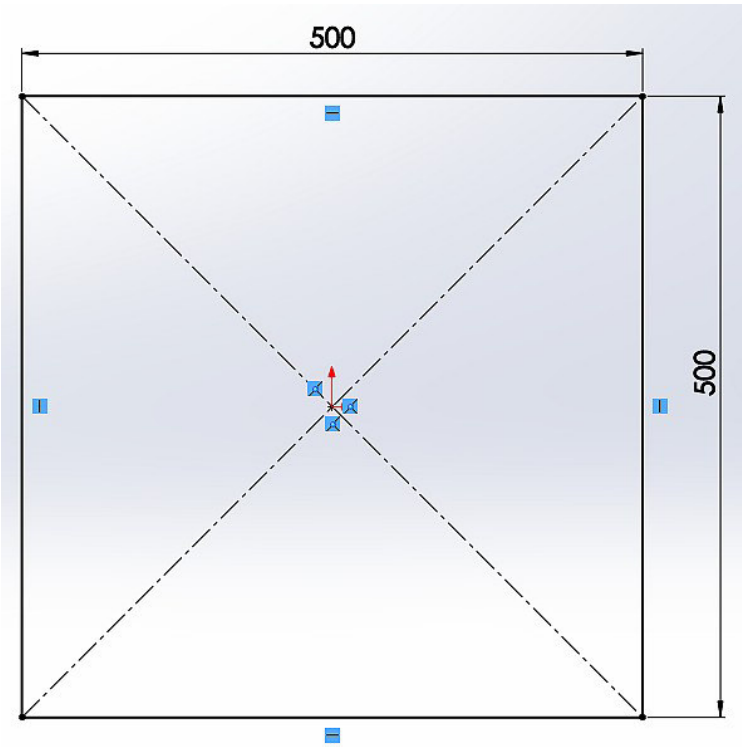
ACTIVITE SUPPLEMENTAIRE SOLIDWORKS

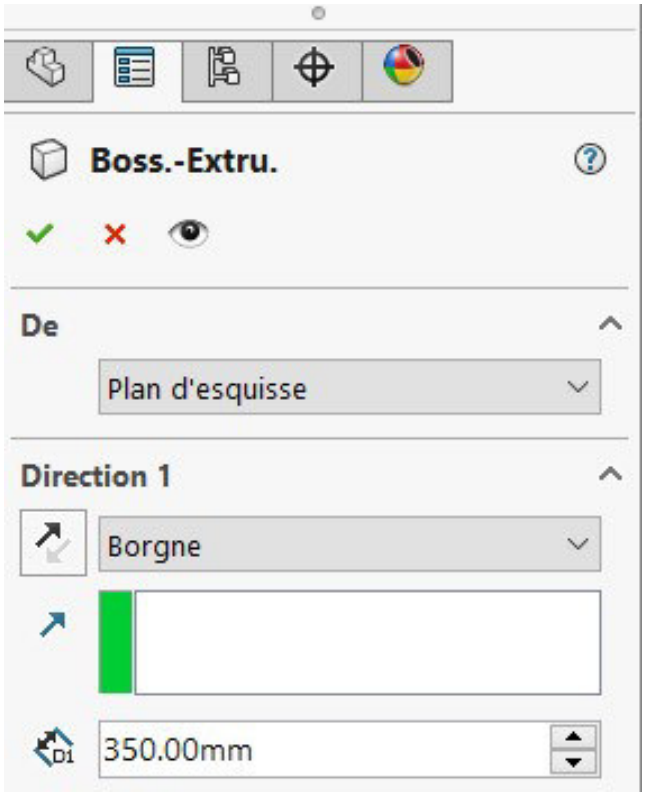
"activité 3 – boite de rangement"

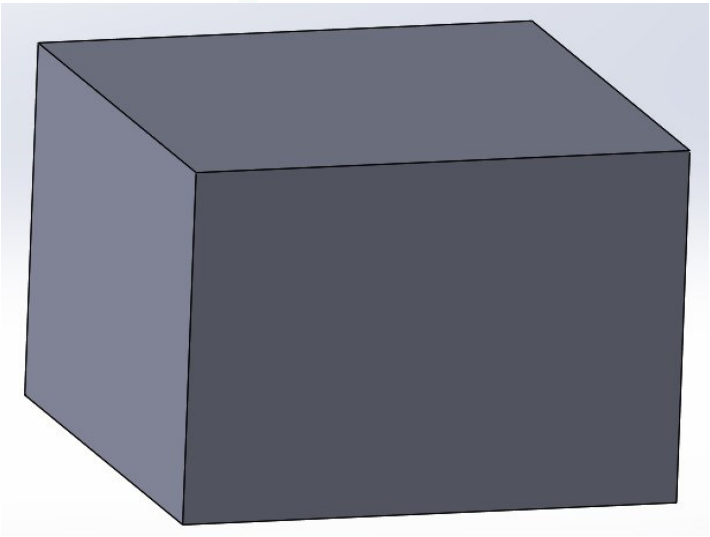
objectifs :

1. *suivre les consignes de conception*
2. *concevoir sur solidworks une boite de rangement*
3. *savoir utiliser les outils esquisse,extrusion,coque*

1/ conception d'une boite de rangement







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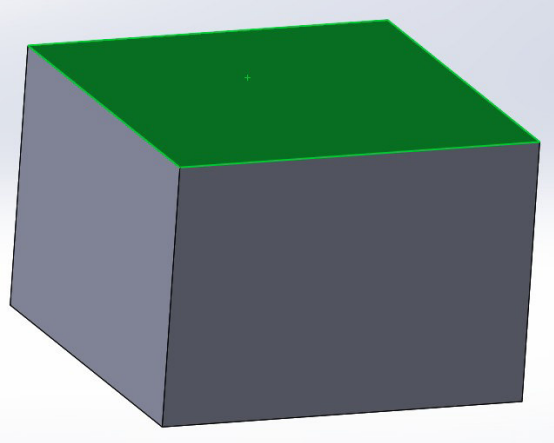
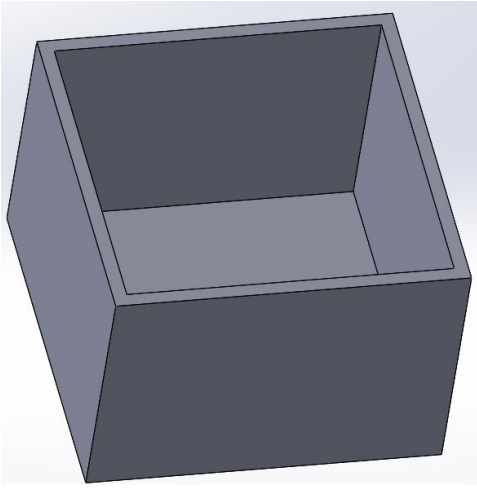
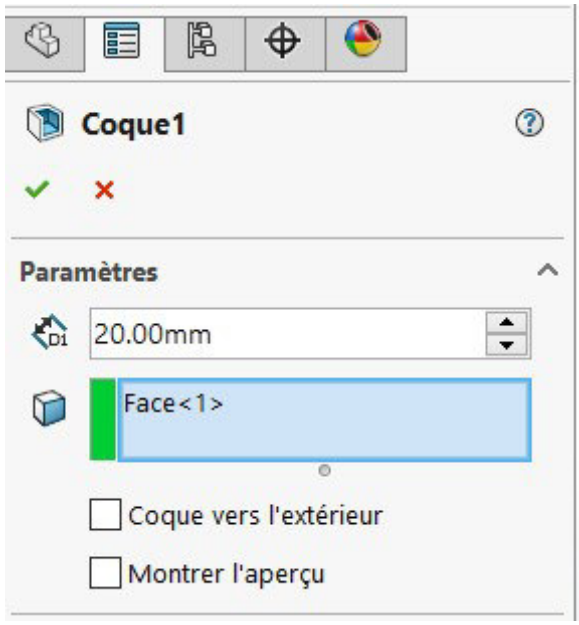
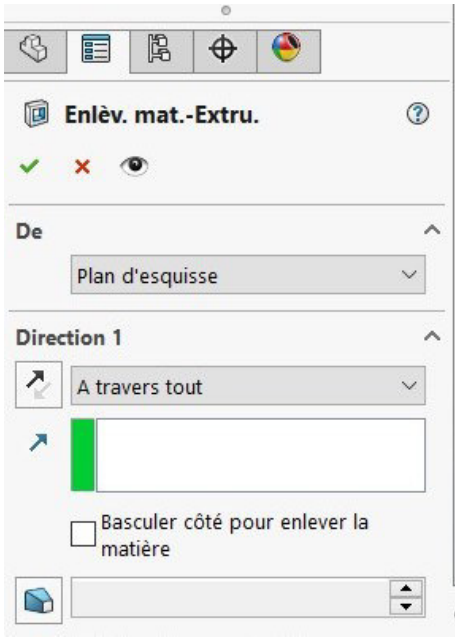
selectionner la face supérieure puis cliquer sur fonction coque

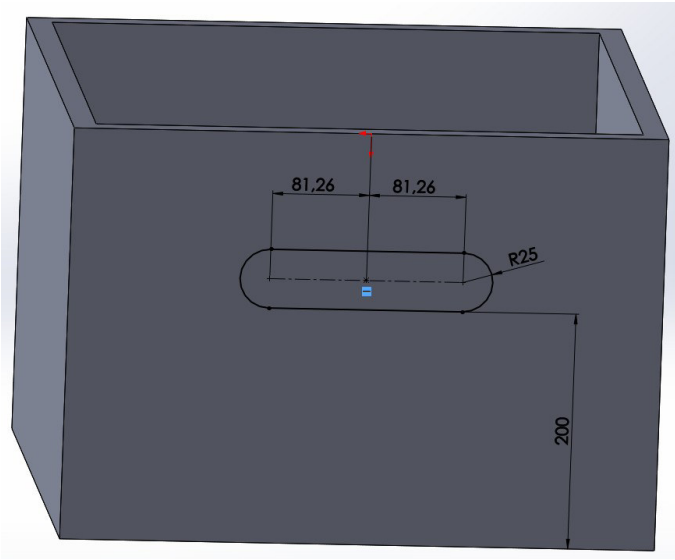


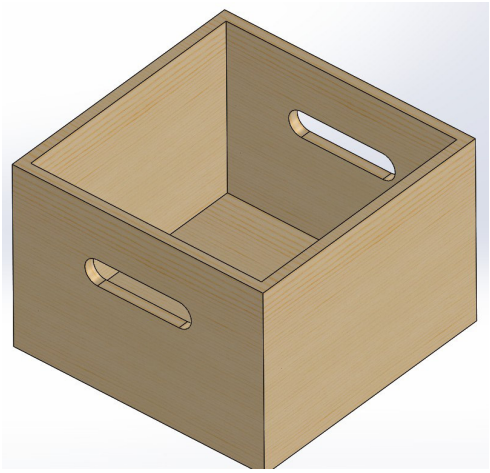






réalisation de la poignet





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ACTIVITE SUPPLEMENTAIRE SOLIDWORKS

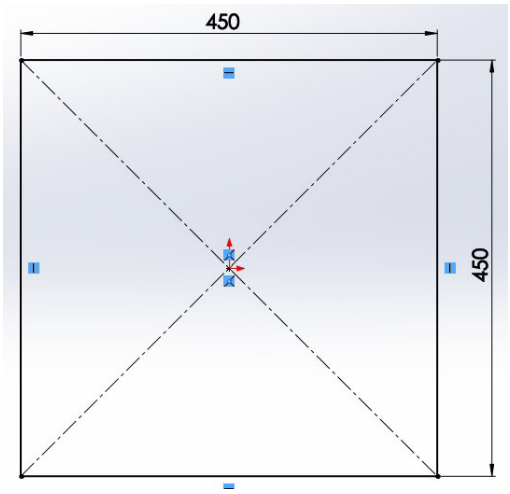
"activité 4 – mini bibiliothèque "

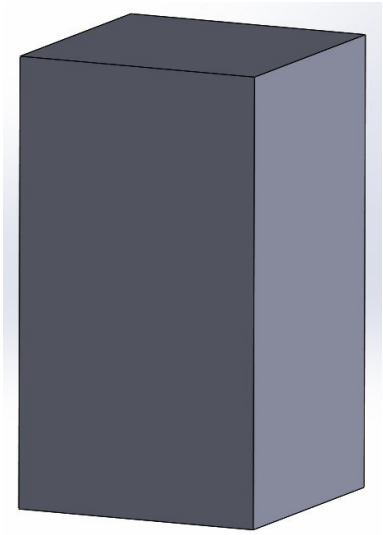
objectifs :

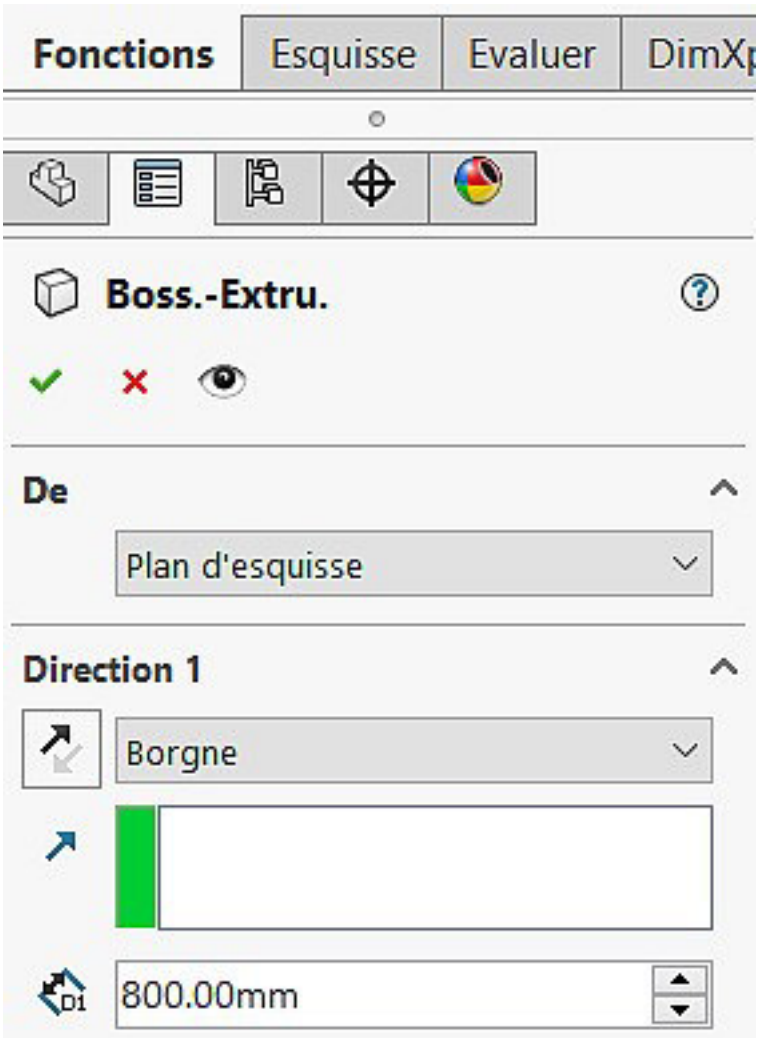
1. *suivre les consignes de conception*
2. *concevoir sur solidworks une mini-bibilothèque*
3. *savoir utiliser les outils extrusion,enlèvement de matière*

1/ conception d'une mini-bibliothèque

- conception de la forme générale







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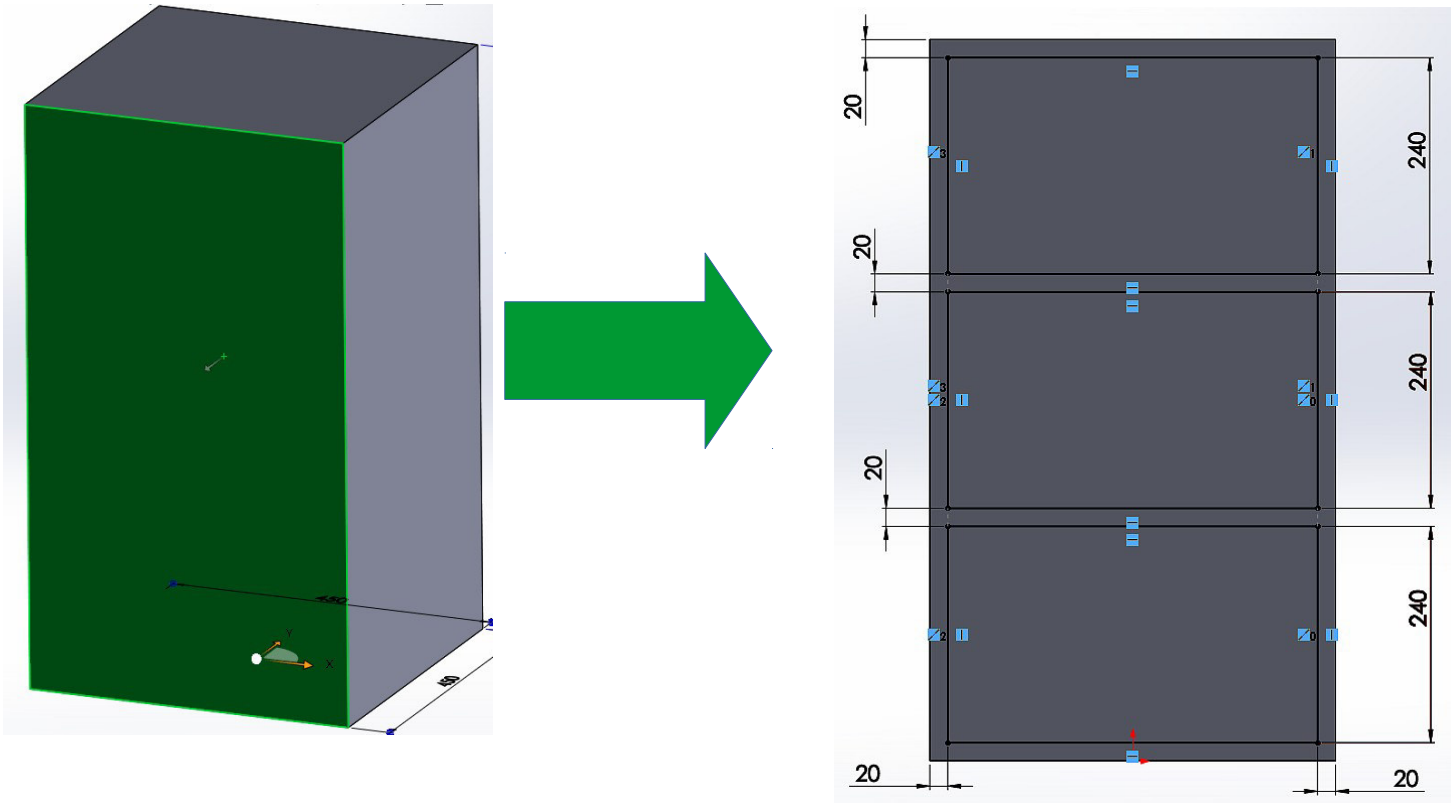
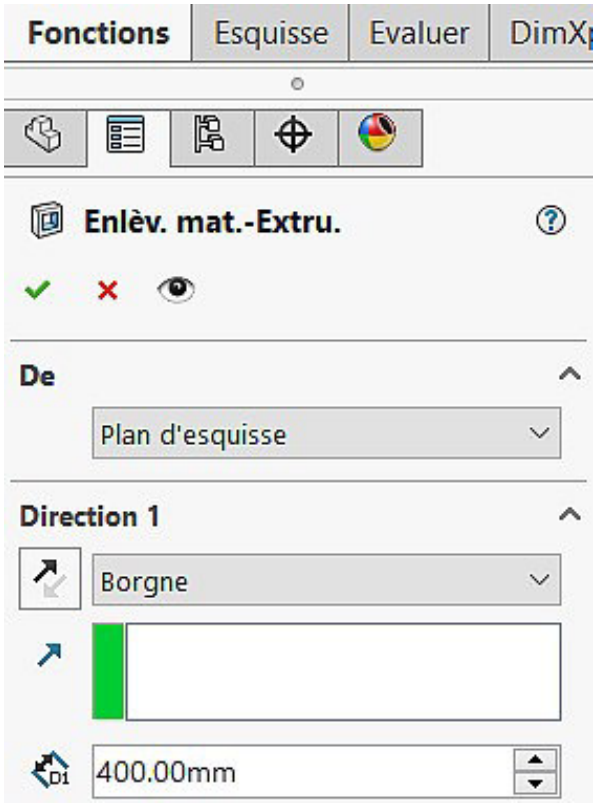
- **conception des tiroirs**

esquisser sur la face l'esquisse suivante



cliquer ensuite sur la fonction enlèvement de matière



MINI-BIBLIOTHEQUE

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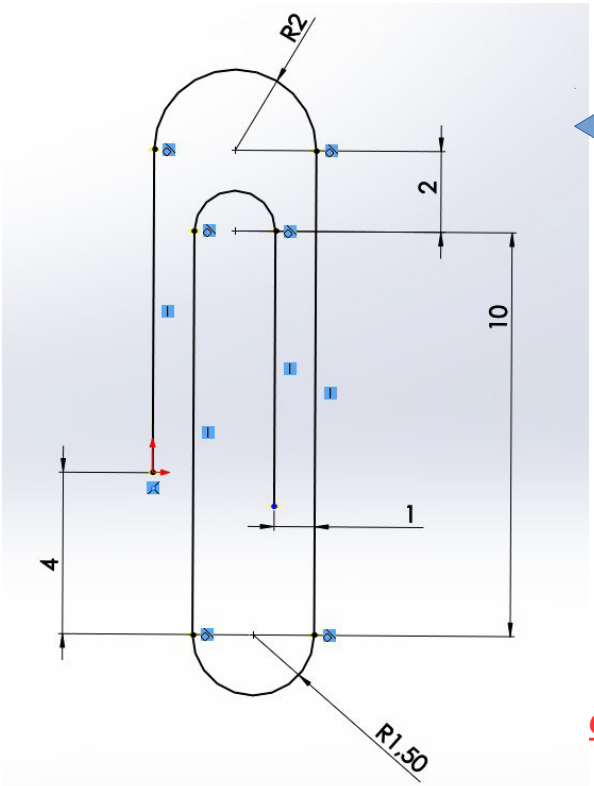
ACTIVITE SUPPLEMENTAIRE SOLIDWORKS

"activité 5 – trombone pour attacher "

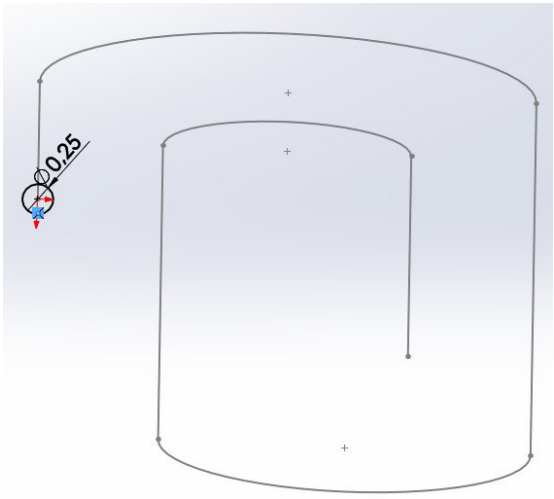
objectifs :

1. suivre les consignes de conception
2. concevoir sur solidworks un trombone pour attacher feuilles
3. savoir utiliser les outils esquisse,base/bossage balayé

1/ conception d'un trombone



1ère esquisse à réaliser en plan de face



esquisse 2 à réaliser en plan de dessus

il faut ensuite cliquer sur la fonction base/bossage balayé

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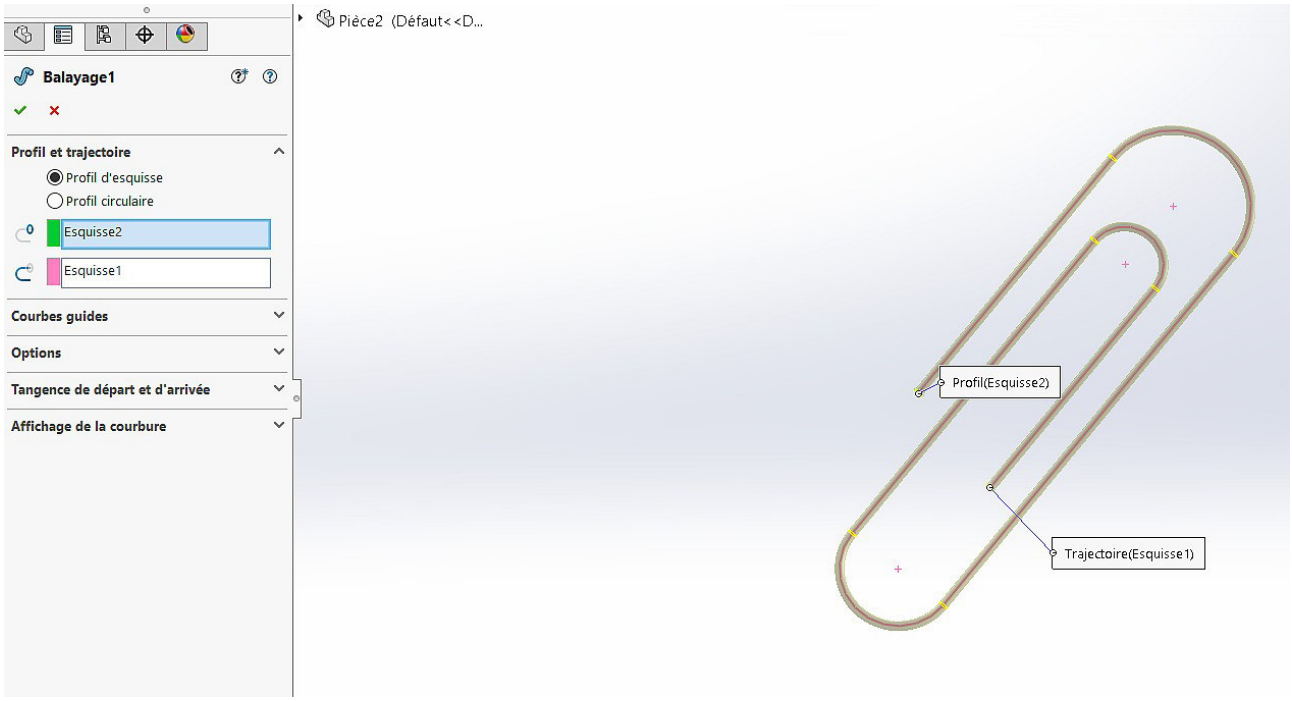
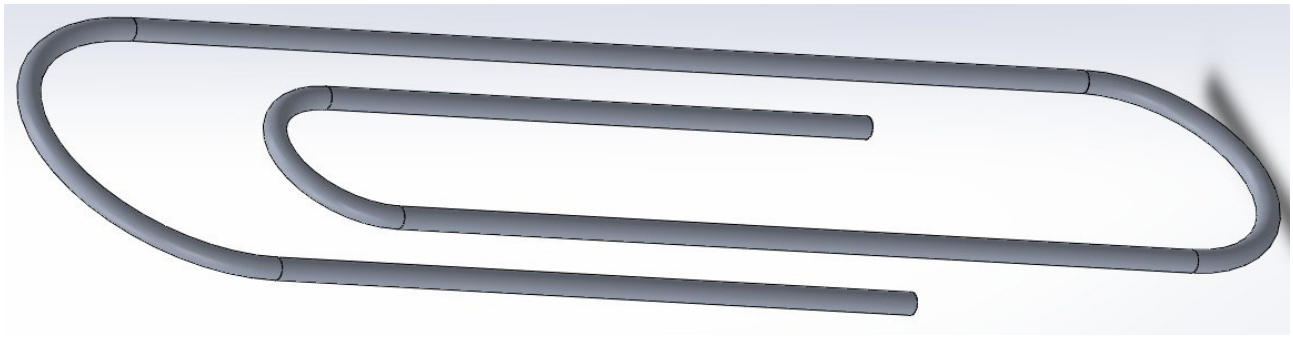
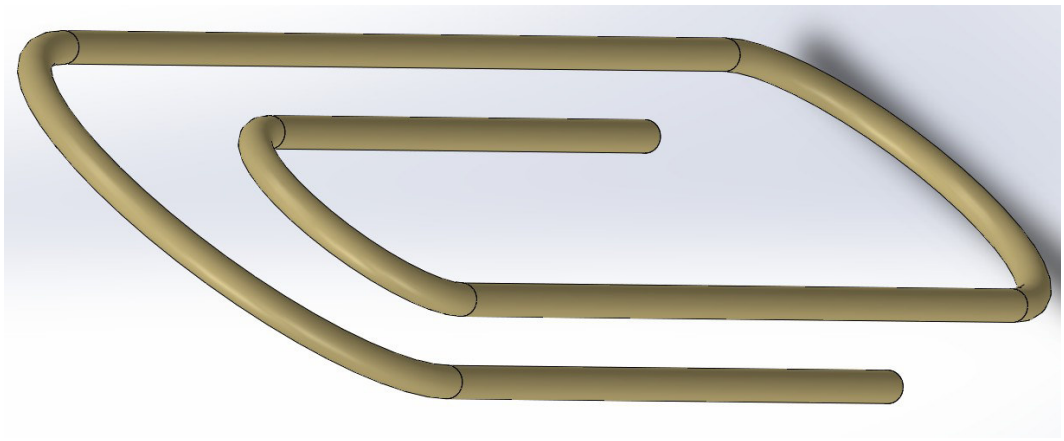


RENDU FINAL DU TROMBONE D'ATTACHE FEUILLES





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