Language="VBSCRIPT"

```vbscript
Sub CATMain()
    Set partDocument1 = CATIA.ActiveDocument
    Set selection1 = partDocument1.Selection
    selection1.Clear
    Set part1 = partDocument1.Part
    Set bodies1 = part1.Bodies
    Set body1 = bodies1.Item("Outil_a_utiliser")
    selection1.Add body1
    selection1.Copy
    Set partDocument1 = CATIA.ActiveDocument    'not necessary
    Set selection2 = partDocument1.Selection
    selection2.Clear
    selection2.Add part1
    selection2.Paste

    Set hybridShapeFactory1 = part1.HybridShapeFactory
    Set hybridShapeDirection1 = hybridShapeFactory1.AddNewDirectionByCoord(1.000000, 0.000000, 0.000000)
    Set shapeFactory1 = part1.ShapeFactory
    Set translate1 = shapeFactory1.AddNewTranslate2(5.000000)
    Set hybridShapeTranslate1 = translate1.HybridShape
    hybridShapeTranslate1.VectorType = 0
    hybridShapeTranslate1.Direction = hybridShapeDirection1
    part1.InWorkObject = hybridShapeTranslate1
    part1.Update
    Set body2 = bodies1.Item("Piece_a_usiner")
    part1.InWorkObject = body2
    Set body3 = bodies1.Item("Outil_a_utiliser")
    Set remove1 = shapeFactory1.AddNewRemove(body3)
    part1.Update

    Set partDocument1 = CATIA.ActiveDocument    'not necessary
    Set selection3 = partDocument1.Selection
    selection3.Clear
    selection3.Add body1
    selection3.Copy
    Set partDocument1 = CATIA.ActiveDocument    'not necessary
    Set selection4 = partDocument1.Selection
    selection4.Clear
    selection4.Add part1
    selection4.Paste

    Set hybridShapeDirection2 = hybridShapeFactory1.AddNewDirectionByCoord(1.000000, 0.000000, 0.000000)
    Set translate2 = shapeFactory1.AddNewTranslate2(10.000000)
    Set hybridShapeTranslate2 = translate2.HybridShape
    hybridShapeTranslate2.VectorType = 0
    hybridShapeTranslate2.Direction = hybridShapeDirection2
    part1.InWorkObject = hybridShapeTranslate2
    part1.Update
    part1.InWorkObject = body2
    Set body4 = bodies1.Item("Outil_a_utiliser")
    Set remove2 = shapeFactory1.AddNewRemove(body4)

    part1.Update
End Sub
```

```vb
Sub CATMain()
    stepNumber = 40

    'le bloc suivant : Récupérer la part ainsi les corps depuis le document activé
    Set partDocument1 = CATIA.ActiveDocument
    Set part1 = partDocument1.Part
    Set bodies1 = part1.Bodies
    'le bloc suivant : Récupérer les deux corps originaux "Outil_a_utiliser" et "Piece_a_usiner"
    Set body1 = bodies1.Item("Outil_a_utiliser")
    Set body2 = bodies1.Item("Piece_a_usiner")

    For step  = 0 To stepNumber-1

        'Dupliquer dans la même part le corps body1 "Outil_a_utiliser"
        Set selection1 = partDocument1.Selection      'définir la première sélection
        selection1.Clear                              'vider la première sélection
        selection1.Add body1                          'ajouter le corps body1 dans la première sélection
        selection1.Copy                               'copier la première sélection
        Set selection2 = partDocument1.Selection      'définir la deuxième sélection
        selection2.Clear                              'vider la deuxième sélection
        selection2.Add part1                          ' ajouter la part dans la deuxième sélection
        selection2.Paste                              'coller ce que l'on a copié avant dans la deuxième sélection

        'Réaliser une translation pour le corps dupliqué suivant l'axe X (1,0,0) de distance 5 mm  * step
        Set hybridShapeFactory1 = part1.HybridShapeFactory
        Set hybridShapeDirection1 = hybridShapeFactory1.AddNewDirectionByCoord(1.000000, 0.000000, 0.000000) 'la direction
        Set shapeFactory1 = part1.ShapeFactory
        Set translate1 = shapeFactory1.AddNewTranslate2(step* 5.000000) ' spécifier la distance de la translation
        Set hybridShapeTranslate1 = translate1.HybridShape
        hybridShapeTranslate1.VectorType = 0
        hybridShapeTranslate1.Direction = hybridShapeDirection1
        part1.Update

        'Réaliser une opération booléenne "Remove" (retirer) sur le corps 2 avec le premier corps  qui s'appelle "Outil_a_utiliser"
        part1.InWorkObject = body2
        Set body3 = bodies1.Item("Outil_a_utiliser")
        Set remove1 = shapeFactory1.AddNewRemove(body3)
        part1.Update

    Next
End Sub
```