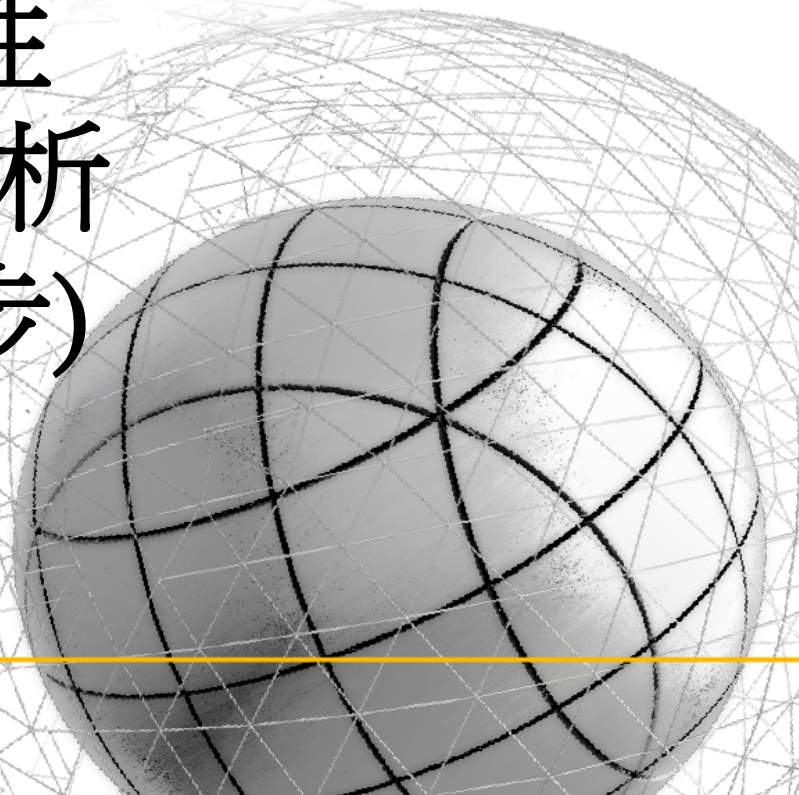
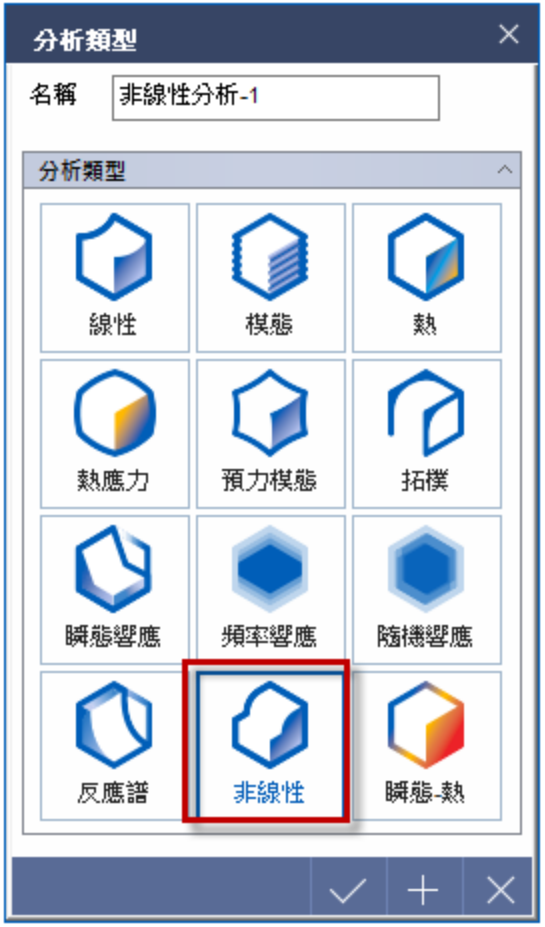
MIDAS

MIDAS MESH FREE

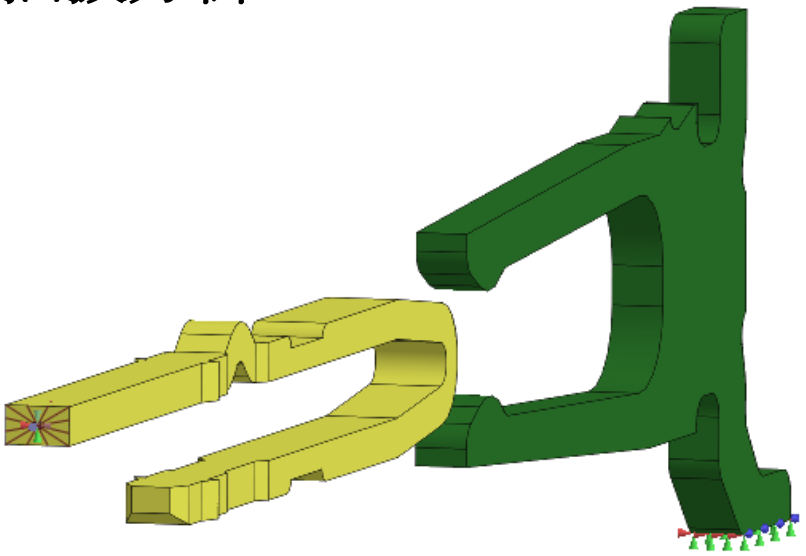
# 接觸非線性
# 端子插拔分析
# (連續分析步)

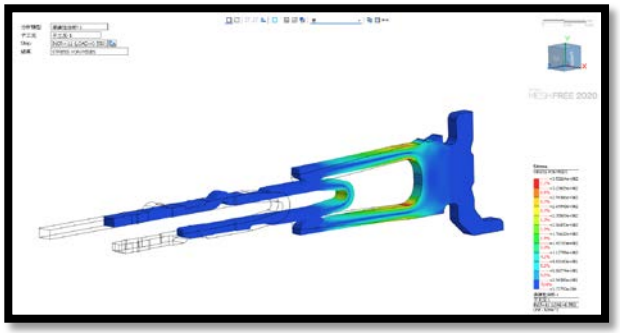


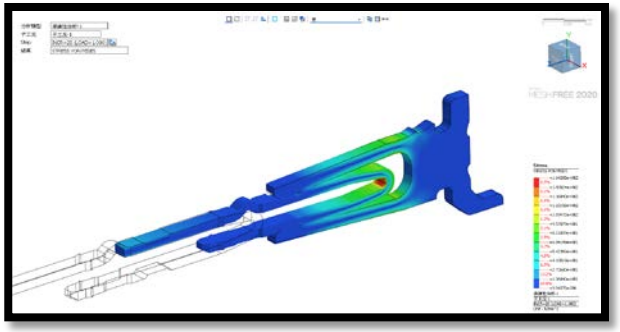
Simple, but Everything.


分析類型
名稱
非線性分析-1
分析類型
線性
模態
熱
熱應力
預力模態
拓樸
瞬態響應
頻率響應
隨機響應
反應譜
非線性
瞬態-熱

# 端子插拔分析



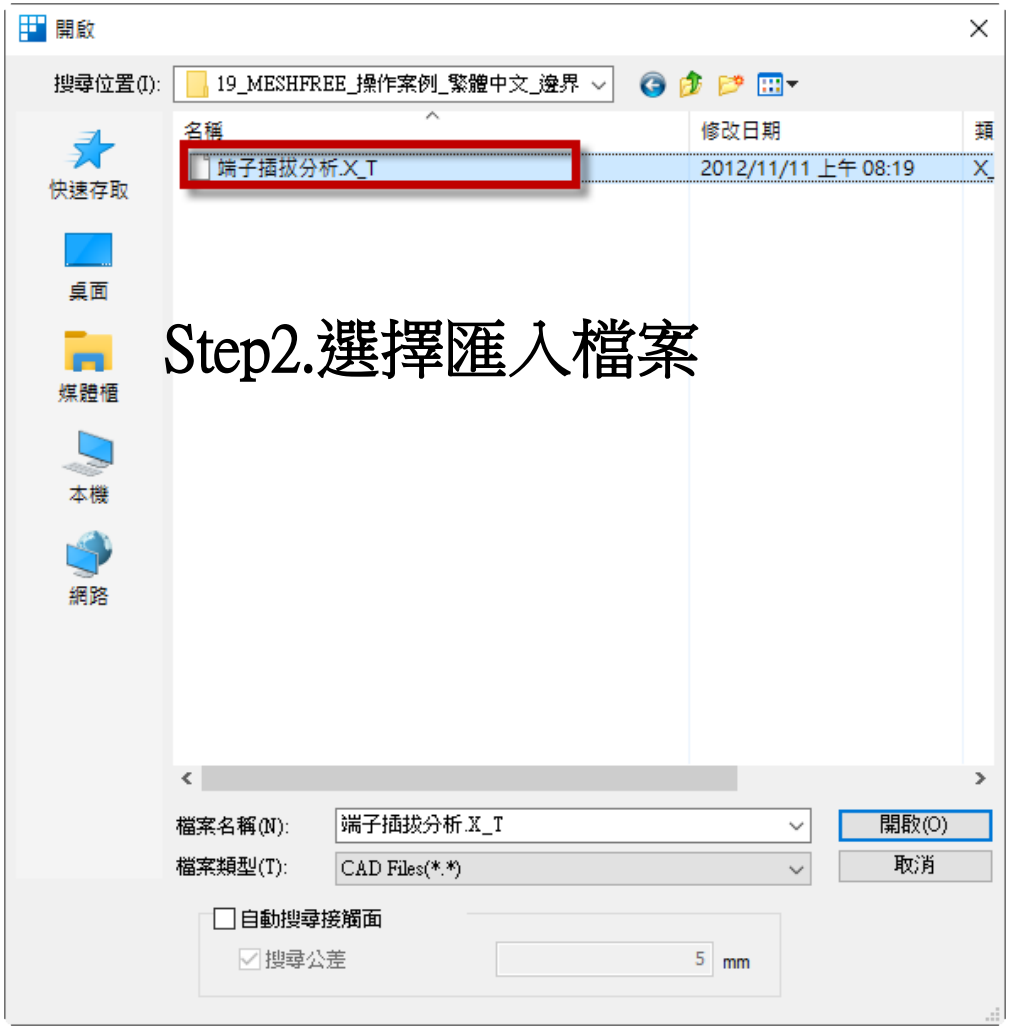





## Step1.匯入3D 模型

## MeshFree支援各類CAD 格式

CAD Files(*.*)
Parasolid (9 - 31) Files (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
ACIS (R1 - 2020 1.0) Files (*.sat;*.sab;*.asat;*.asab)
STEP (AP203, AP214, AP242) Files (*.stp;*.step)
IGES (Up to 5.3) Files (*.igs;*.iges)
Pro-E (16 - Creo 6.0) Files (*.prt;*.prt.*;*.asm;*.asm.*)
CATIA V4 (CATIA 4.1.9 - 4.2.4) Files (*.model;*.exp;*.session)
CATIA V5 (V5R8 - V5-6R2019) Files (*.CATPart;*.CATProduct)
Solid Works (98 - 2020) Files (*.sldprt;*.sldasm)
Unigraphics (11 - NX1847) Files (*.prt)
Inventor Part (V6 - V2020) Files (*.ipt)
Inventor Assembly (V11 - V2020) Files (*.iam)
Solid Edge (V18 - ST11) Files (*.par;*.asm;*.psm)

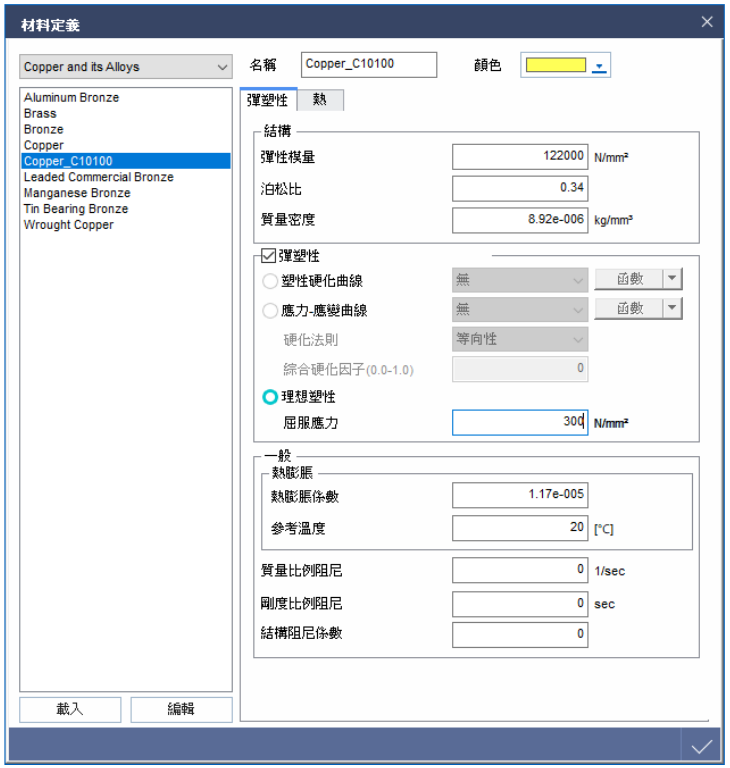


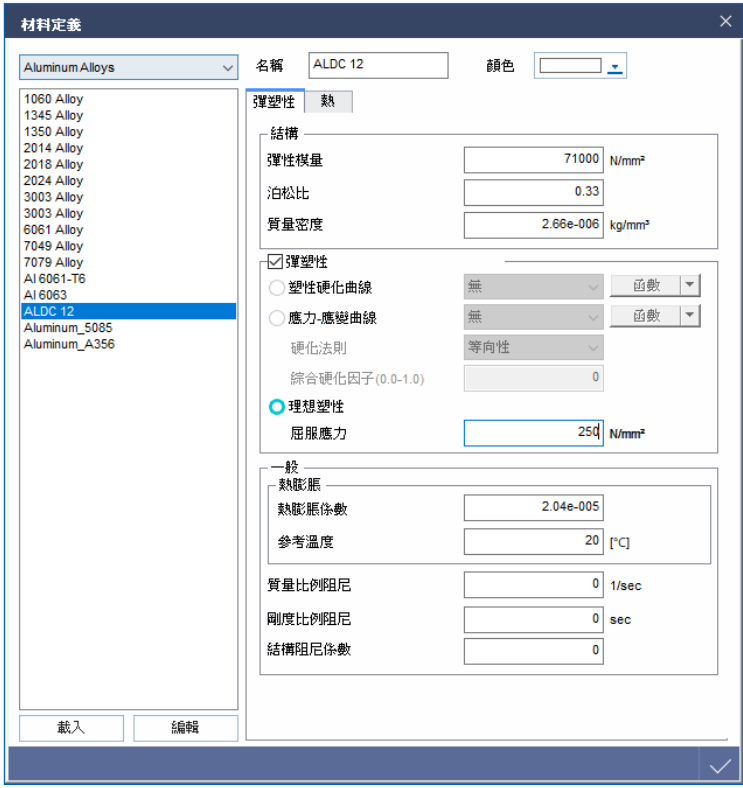

## Step2.選擇匯入檔案

## Step3.關閉自動搜尋接觸面

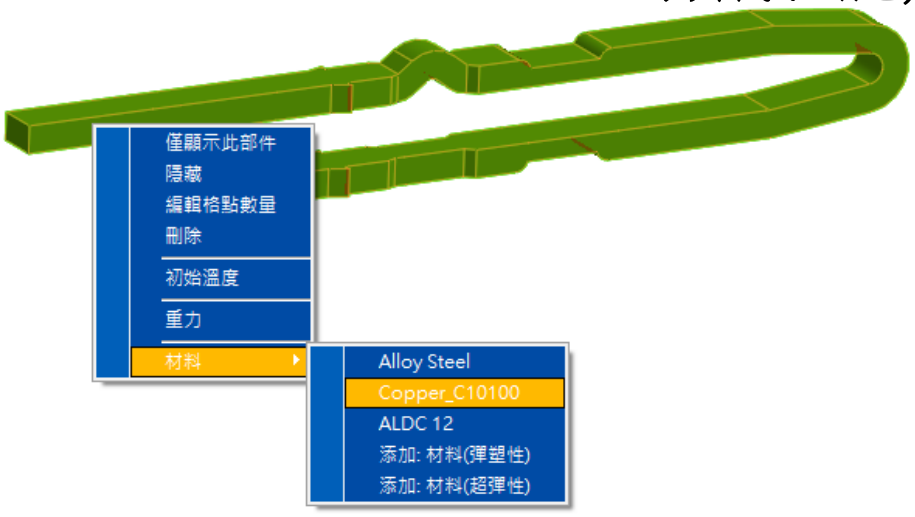

# 新增彈塑性材料

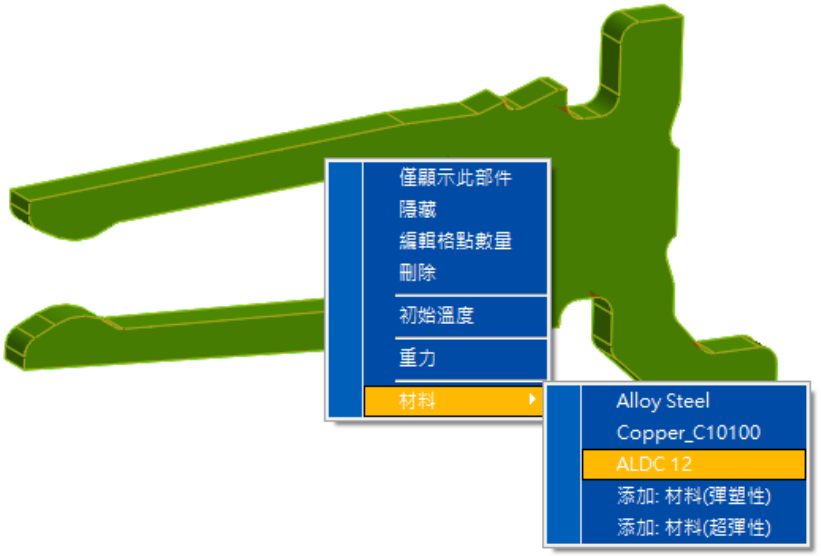
# 新增Copper C10100和ALDC12 (分別指定屈服應力)



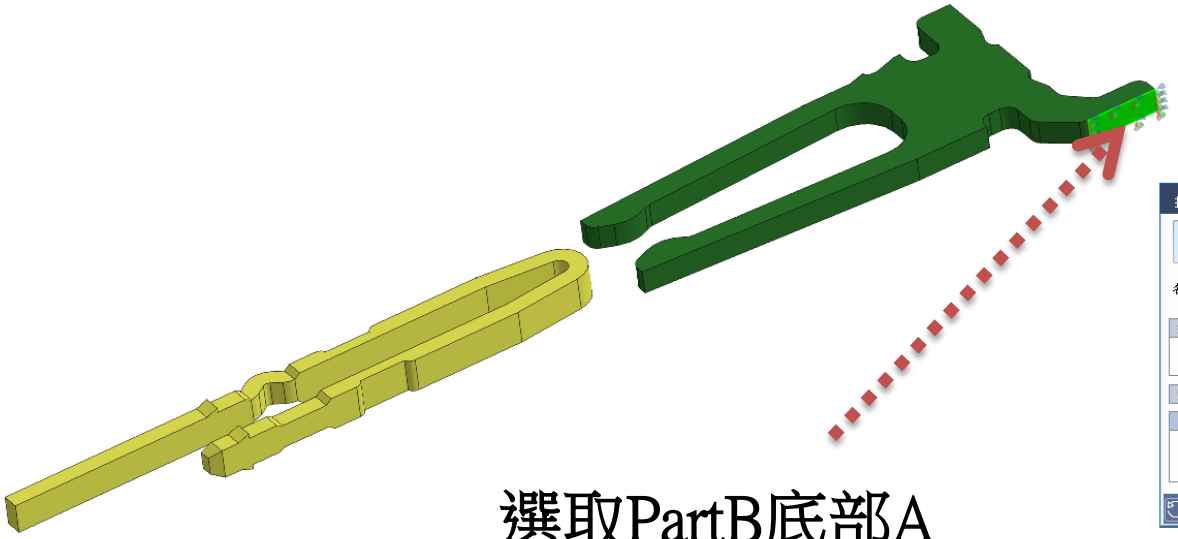
# 滑鼠右鍵,材料指定





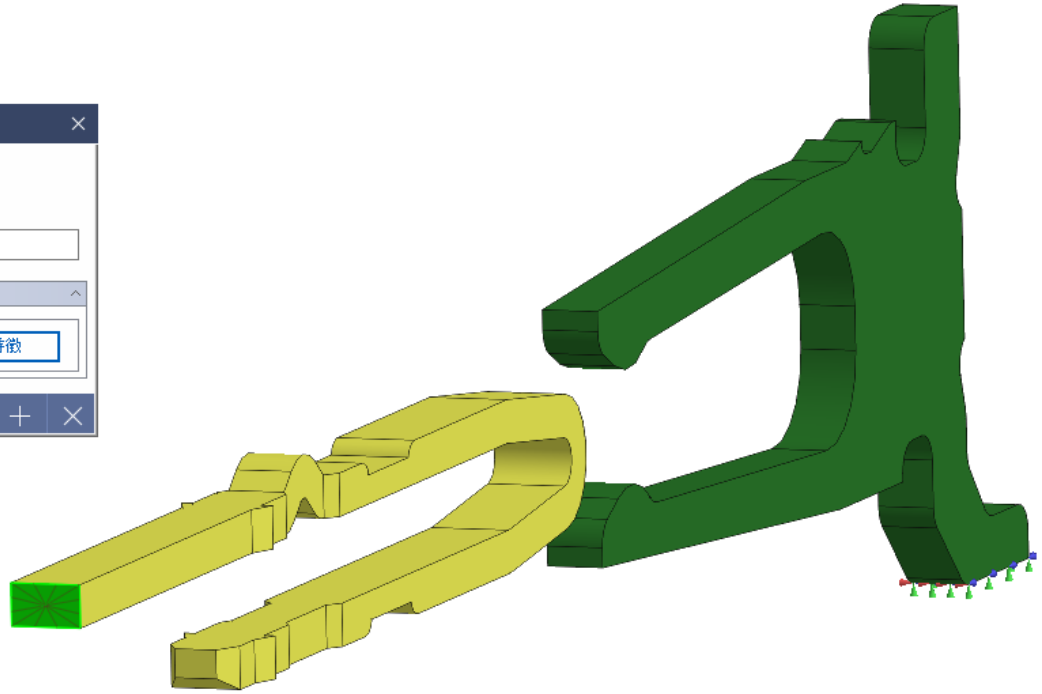
模型樹顯示指定材料


# 選取PartB底部A
# 拘束(TX/TY/TZ)

定義拘束邊界條件。
(對稱模型可以採用對稱條件)

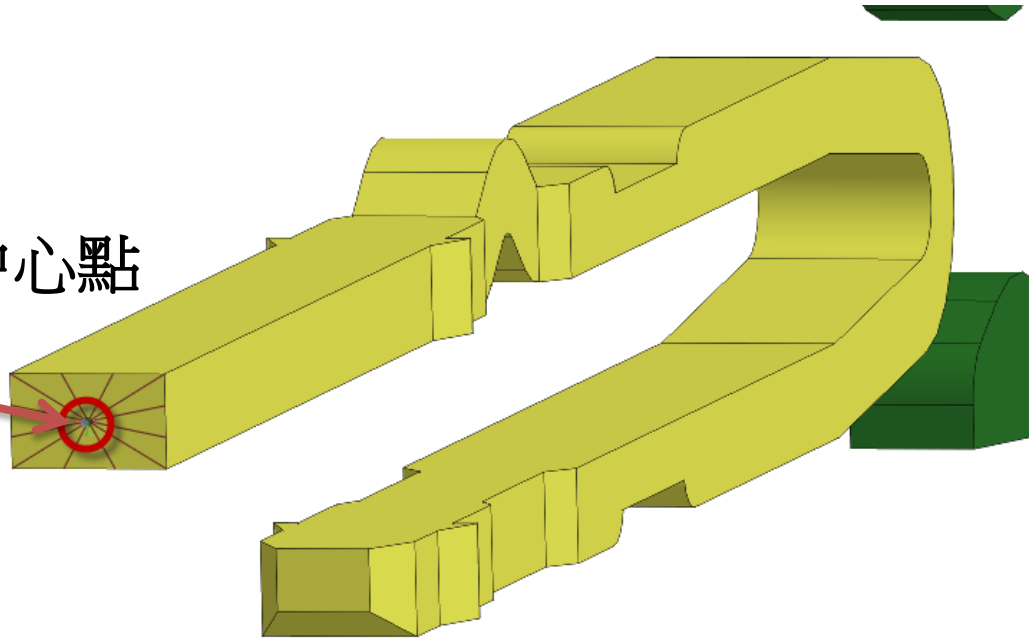




# PartA新增剛性連接


# 採用點特徵拘束


選取剛性連間中心點



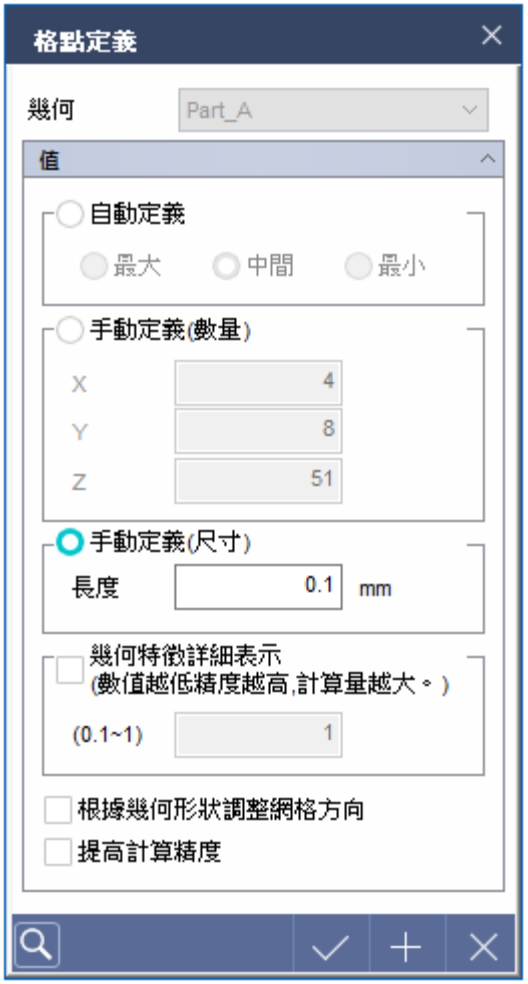
固定TX/TY/RX/RY/RZ

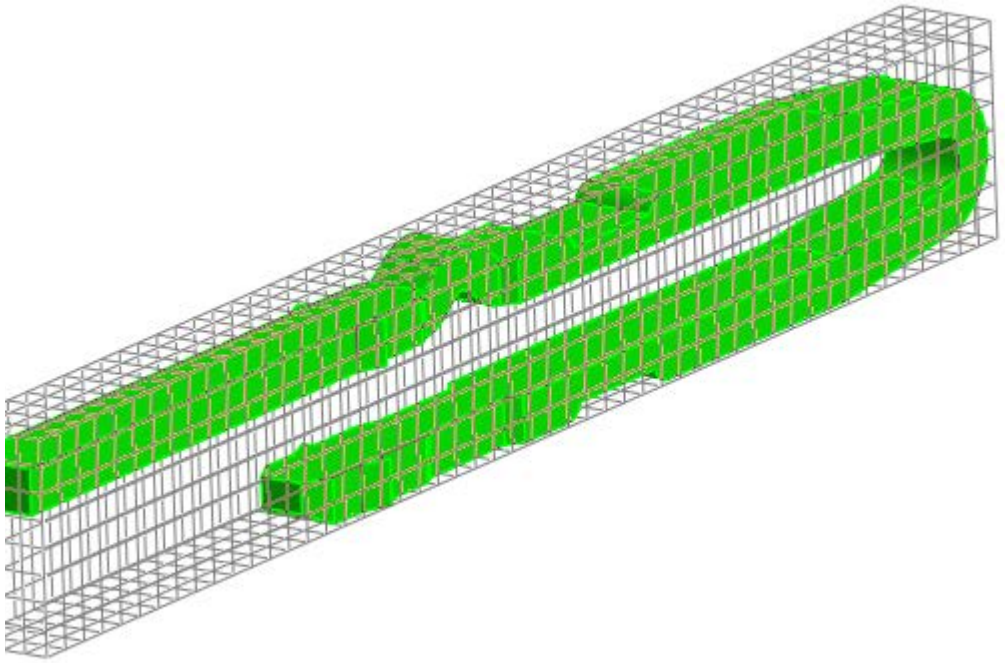

# 施加二組位移條件

## 隱藏PartB/PartA

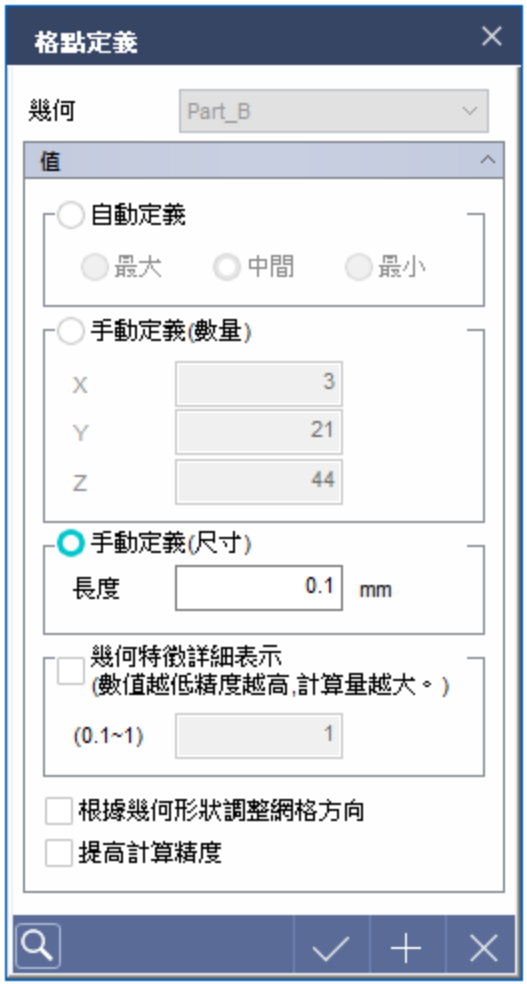


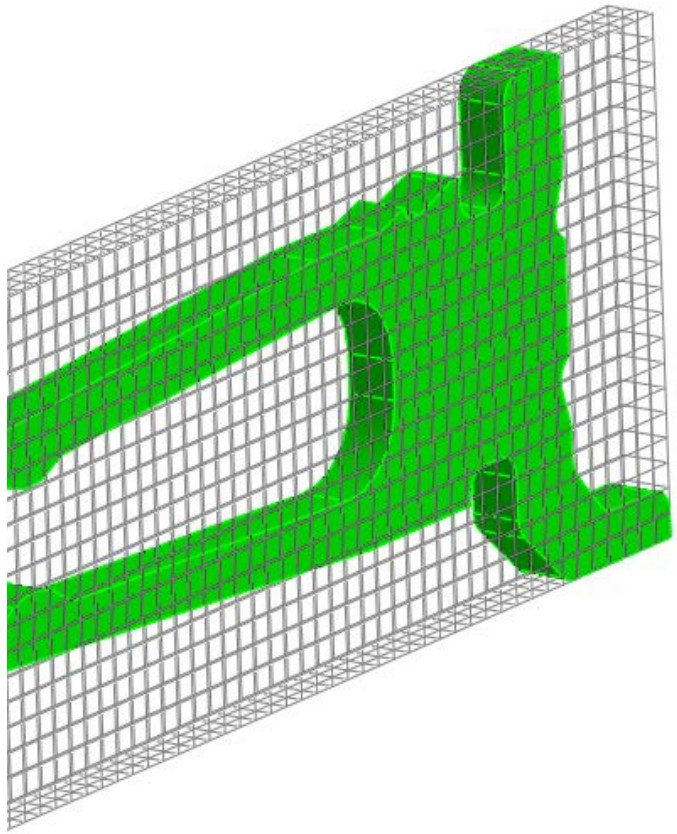
# 分別定義格點尺寸(Part A)



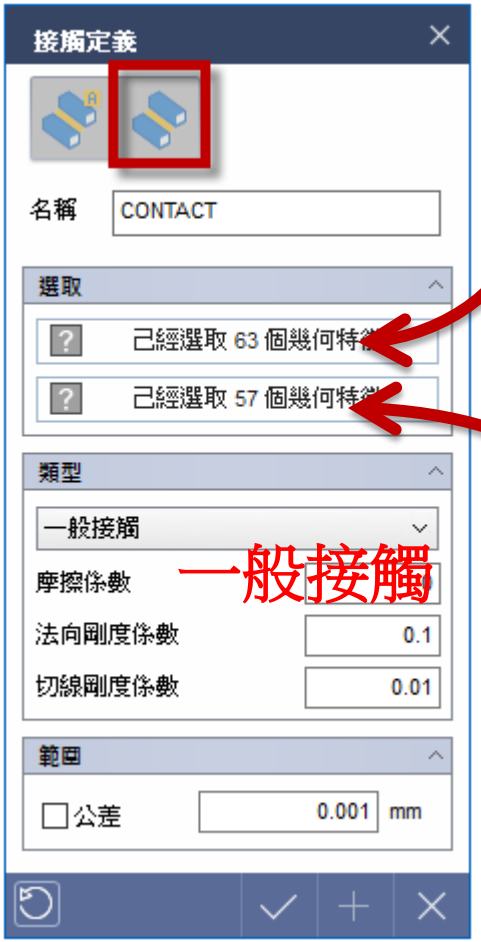



# 分別定義格點尺寸(Part B)



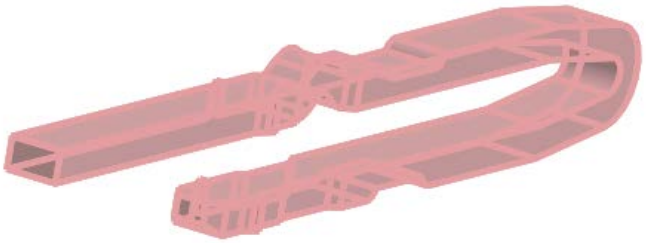
# 手動接觸定義

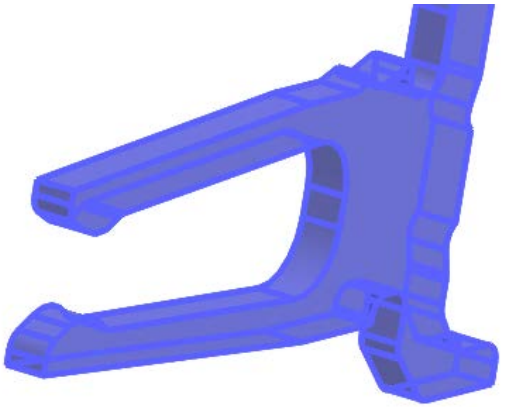



Master

Slave

Part A

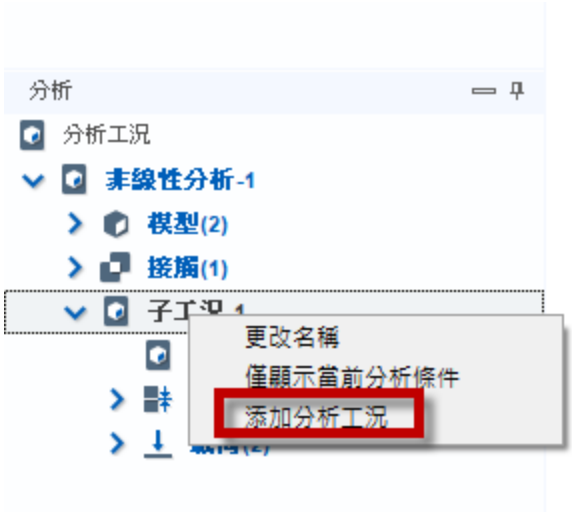




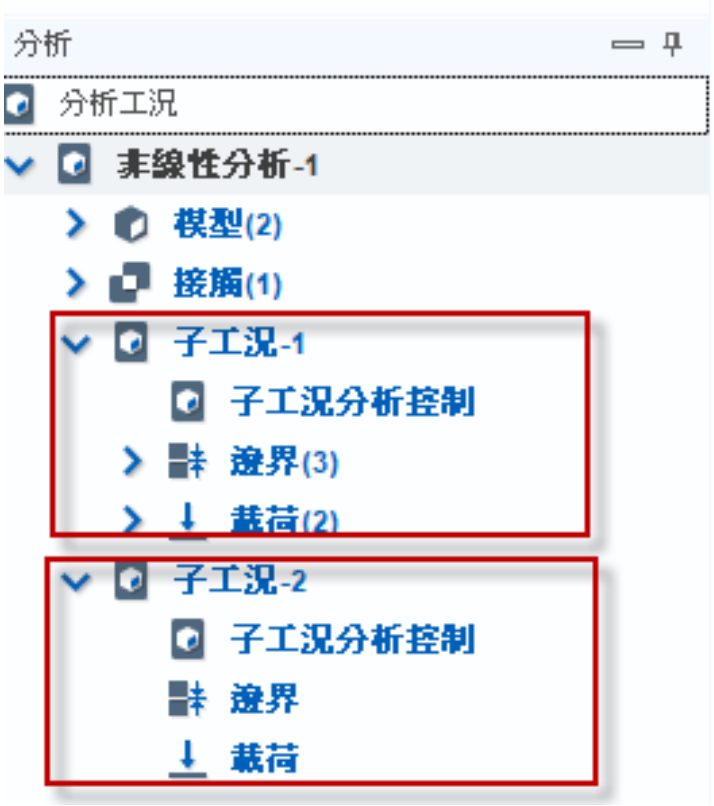
Part B

註:接觸範圍指的是允許穿透間距

# 新增連續分析步

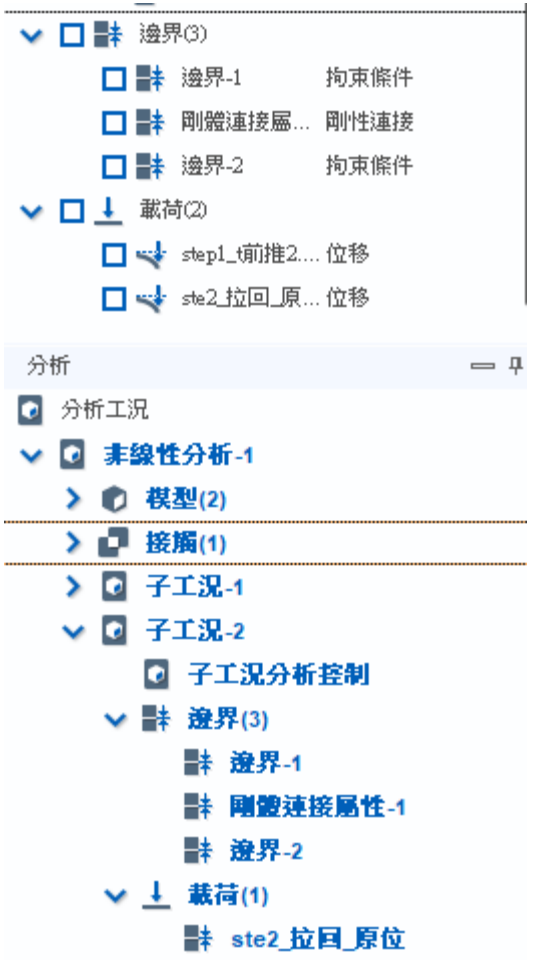



滑鼠右鍵,添加分析工況



# 編輯連續分析步


子工況-1:刪除STEP2載荷




子工況-2:拖曳邊界和STEP2載荷

# 分析工況控制(子工況-1/子工況-2相同)


註:無網格法接觸不容易收斂,適度調高收斂準則可以加速收斂


記憶體大小
1.計算速度
2.分析準確性