

FEA NX_標準教學系列

鋼骨結構-線性分析
多載荷類型&焊接接觸

台灣邁達斯

組合件-連接/接觸方式

視作單一元件

方式1. CAD模型進行布林運算，將多個元件合併成單一元件。

方式2. 元素共節點。

接觸條件

類型1. 焊接接觸

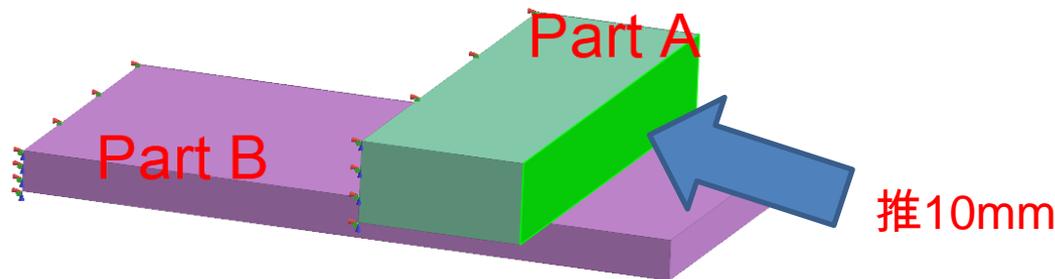
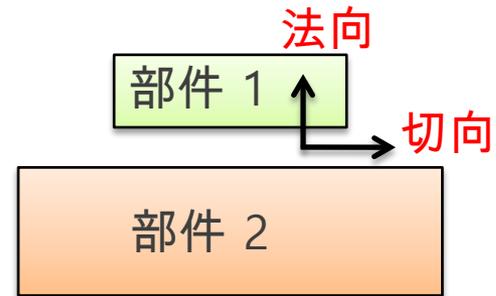
雙向固定

類型2. 滑動接觸：

單向滑動(允許切線方向些許滑動)

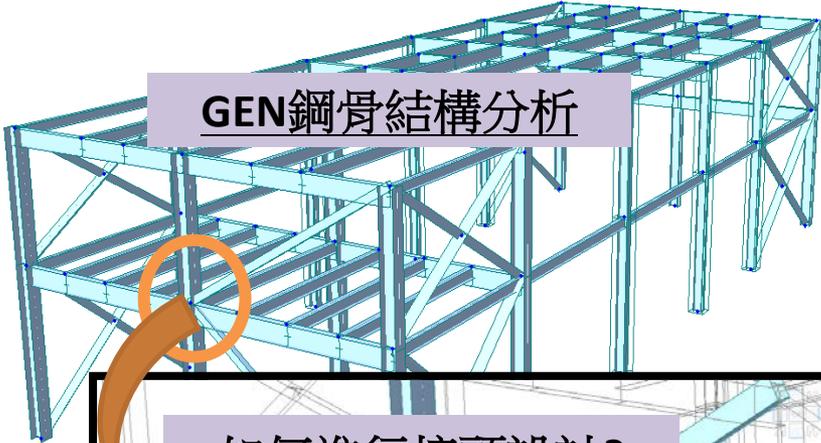
類型3. 一般接觸

雙向滑動(線性分析不支援)

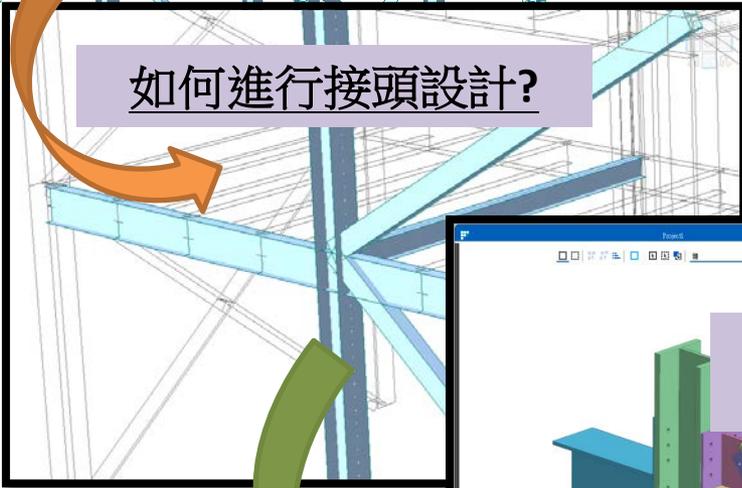


MIDAS-整合分析

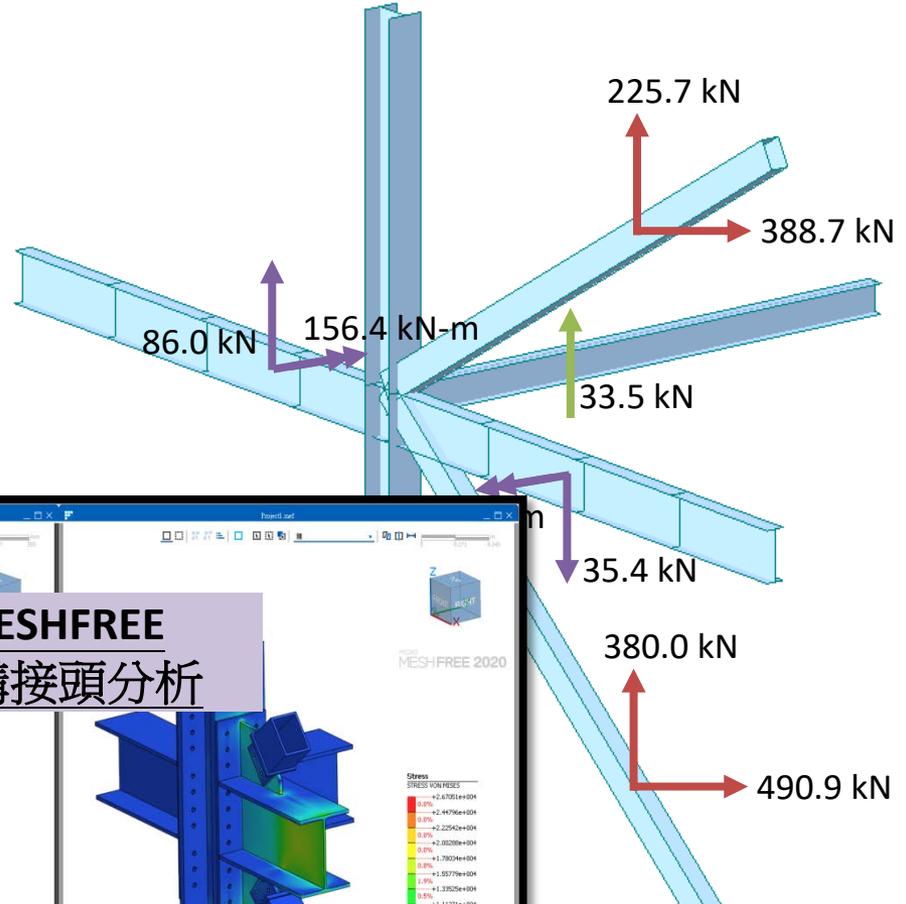
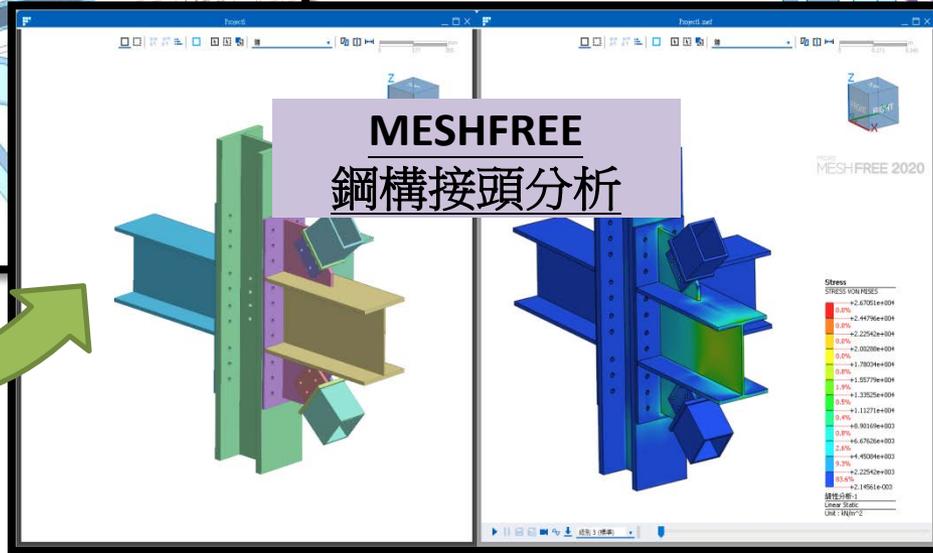
GEN鋼骨結構分析



如何進行接頭設計?



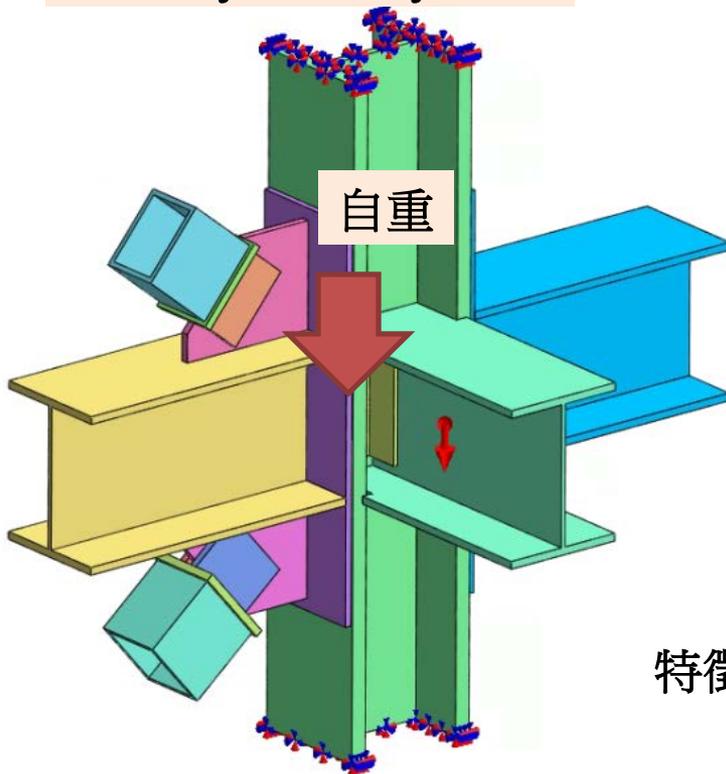
MESHFREE
鋼構接頭分析



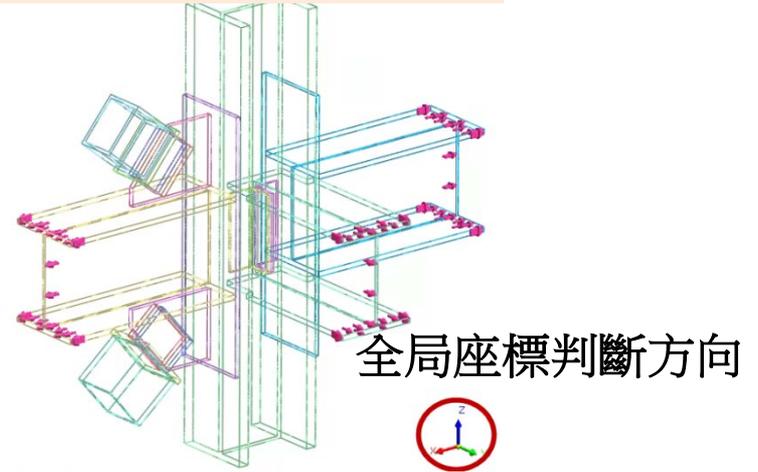
分析說明

材料
Steel/CNS(S)/SS490

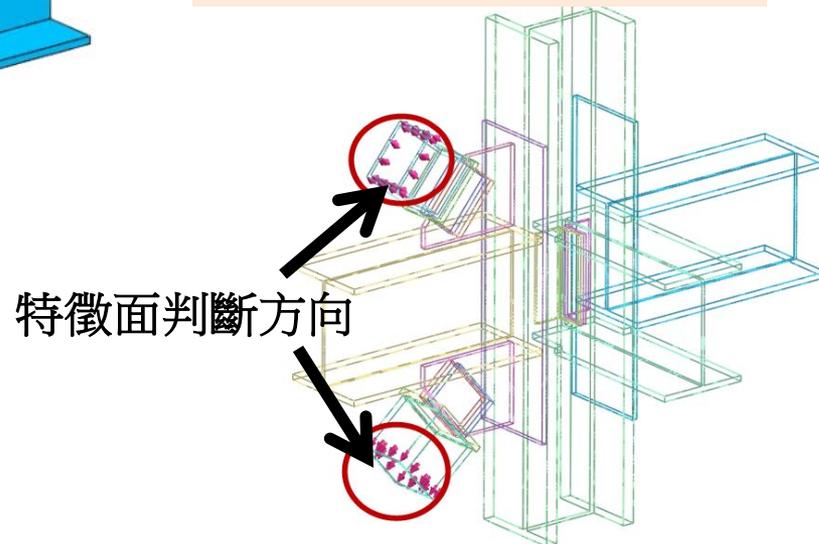
拘束立柱上下面特徵
 $T_x/T_y/T_z/R_x/R_y/R_z$



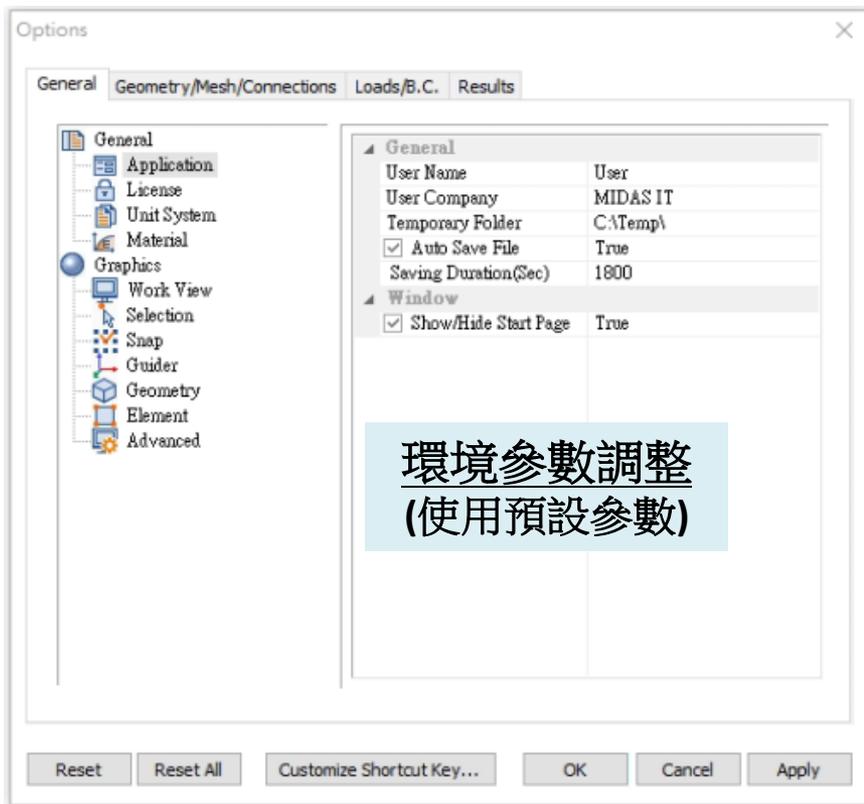
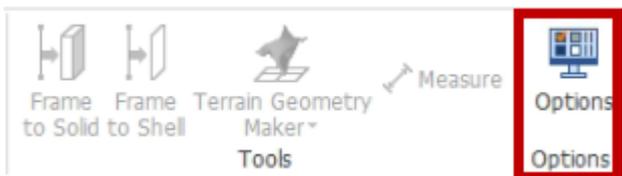
鋼構橫樑施加50000(Nt)



鋼構斜拉端施加25000(Nt)



環境

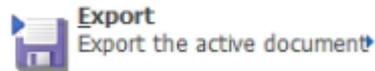


新文件

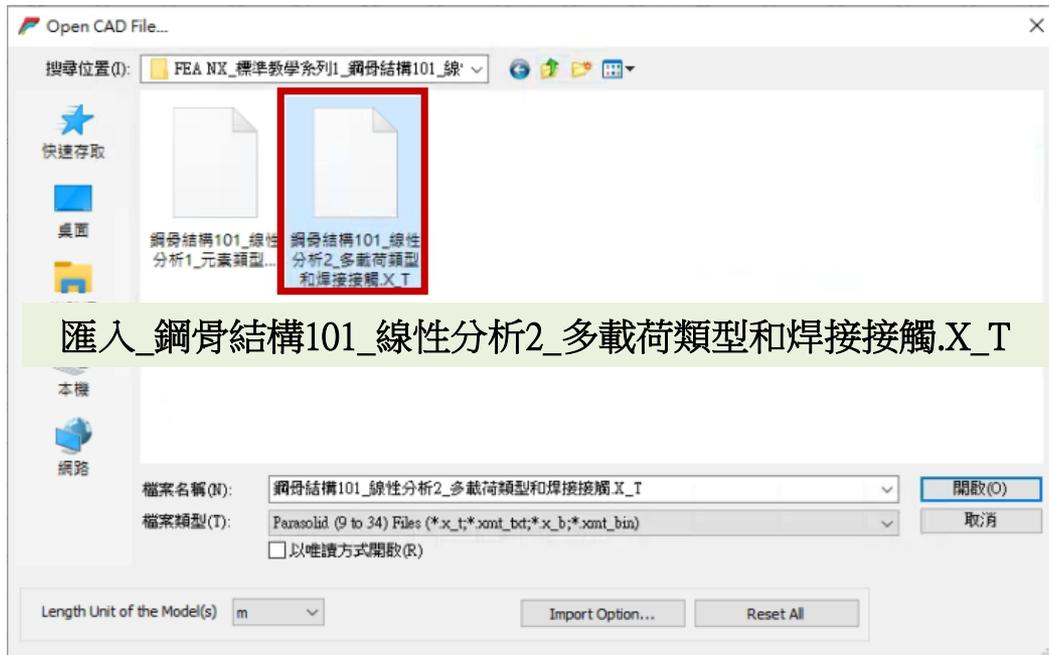


單位使用N/m/J/sec

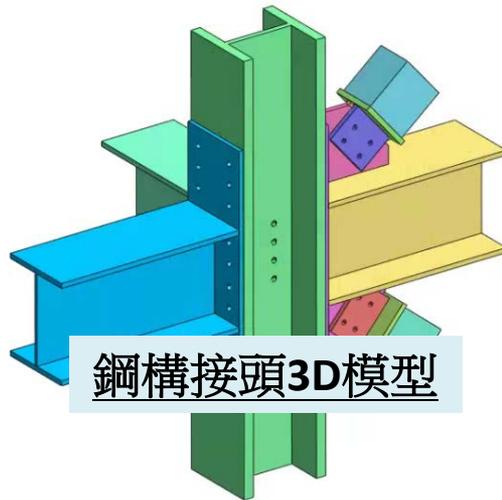
匯入模型



- Import the geometries or FE model in the selected file
- Import CAD File...**
Import the CAD file into a current project
 - DXF 2D (Wireframe)...**
Import the DXF 2D file into a current project
 - DXF 3D (Wireframe)...**
Import the DXF 3D file into a current project
 - DWG (Wireframe)...**
Import the DWG file into a current project
 - midas Mxt...**
Import the midas MXT file.
 - GeoXD Neutral Format File(*.FPN)...**
Import the Neutral File
 - GTS NX Neutral Format...**
Open Neutral File
 - Import Nodal Results(*.txt)**
Import nodal results File

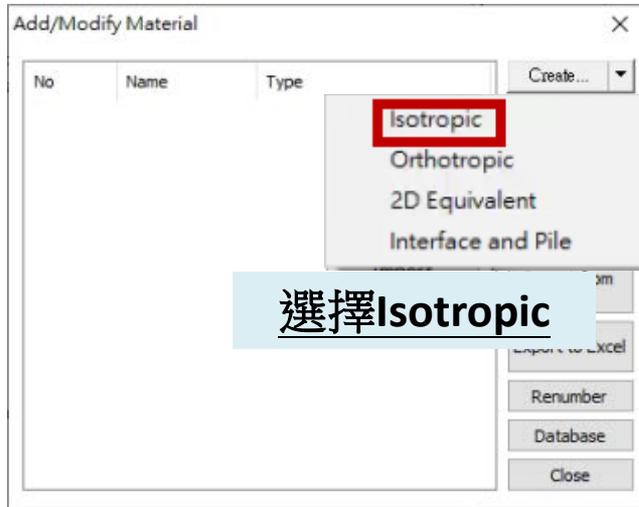
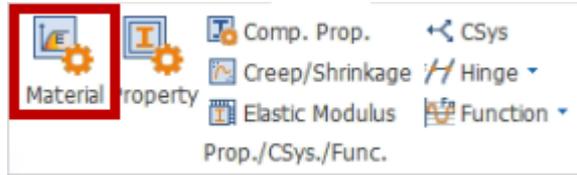


匯入_鋼骨結構101_線性分析2_多載荷類型和焊接接觸.X_T

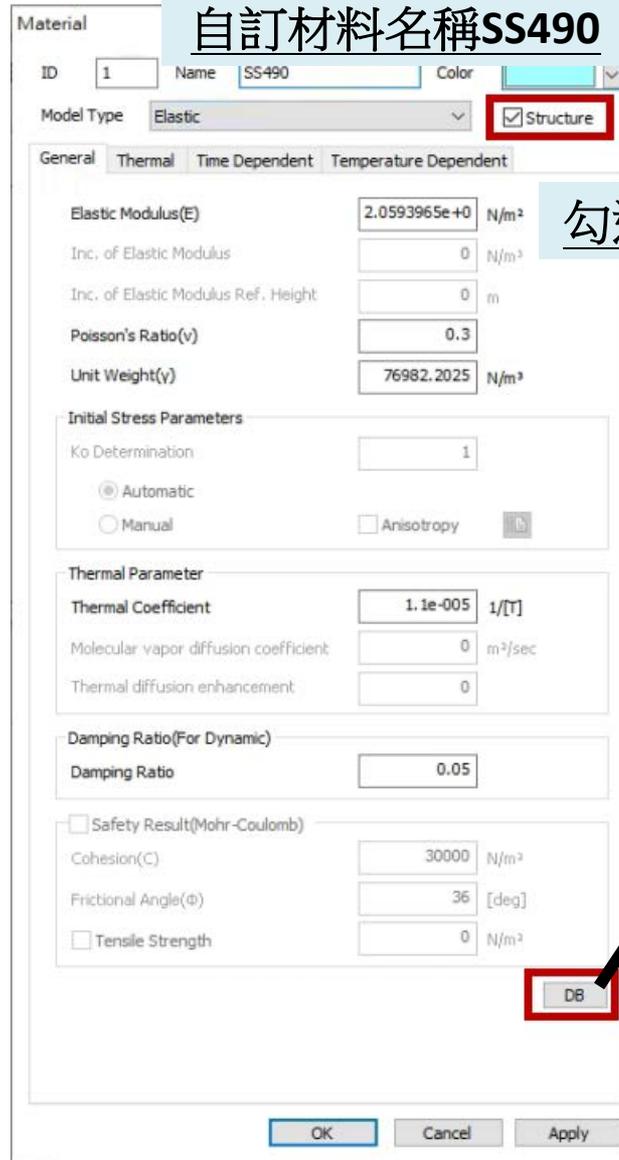


鋼構接頭3D模型

材料



自訂材料名稱SS490



勾選Structure

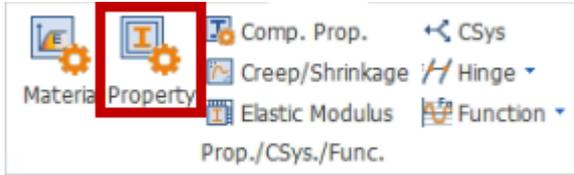
Steel/CNS(S)/SS490



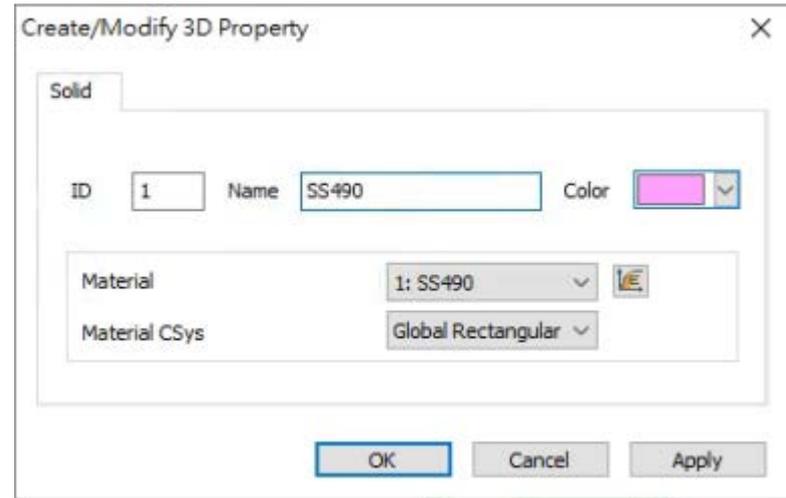
材料庫(混凝土/鋼材)

註:未勾選Structure則視為土壤材料。

屬性



新增SS490 Property



幾何清理

(孔徑特徵)

- Remove Manual
- Repair Shape
- Imprinted Object
- Edges-Domain
- Check Geometry
- Check Duplicate
- Remove/Modify

使用孔徑特徵自動搜索

Model Simplification

Auto Manual

Select Target Object(s)

Search condition

- Hole 0.0519667196 m
- Fillet < 0.0519667196 m
- Small Edge < 0.0101 m
- Small Face < 0.0101 m
- Sliver Face < 0.0101 m
- Spike < 0.0101 m

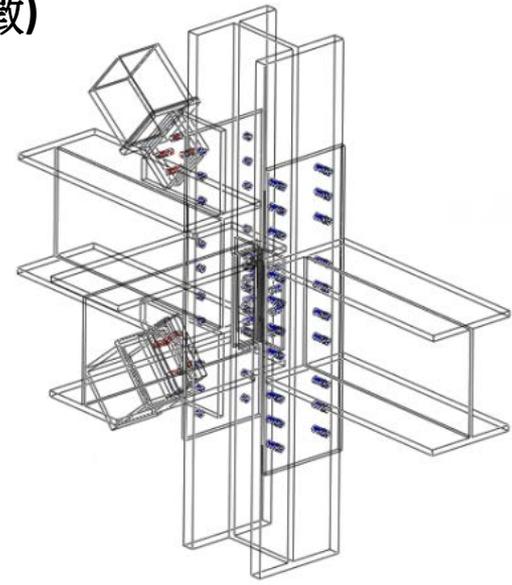
Find

Results (double click to focus on the geometry)

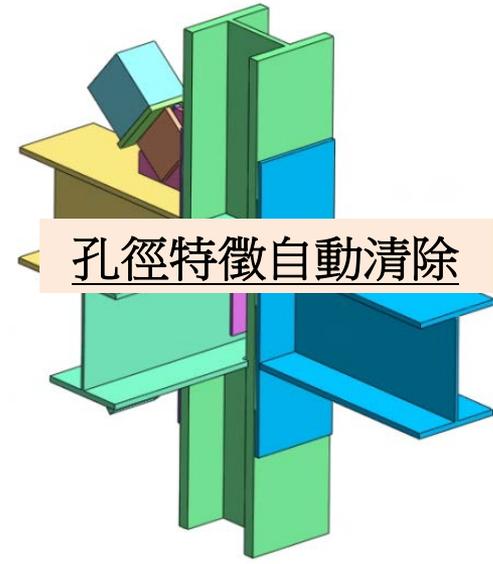
Color	Part Name	Type	Value
✓	Body	Hole	0.009000
✓	Body	Hole	0.009000
✓	Body	Hole	0.009000
✓	Body	Hole	0.009000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000
✓	Body	Hole	0.011000

Select All Unselect All Remove

全部孔徑特徵清除



孔徑特徵自動清除

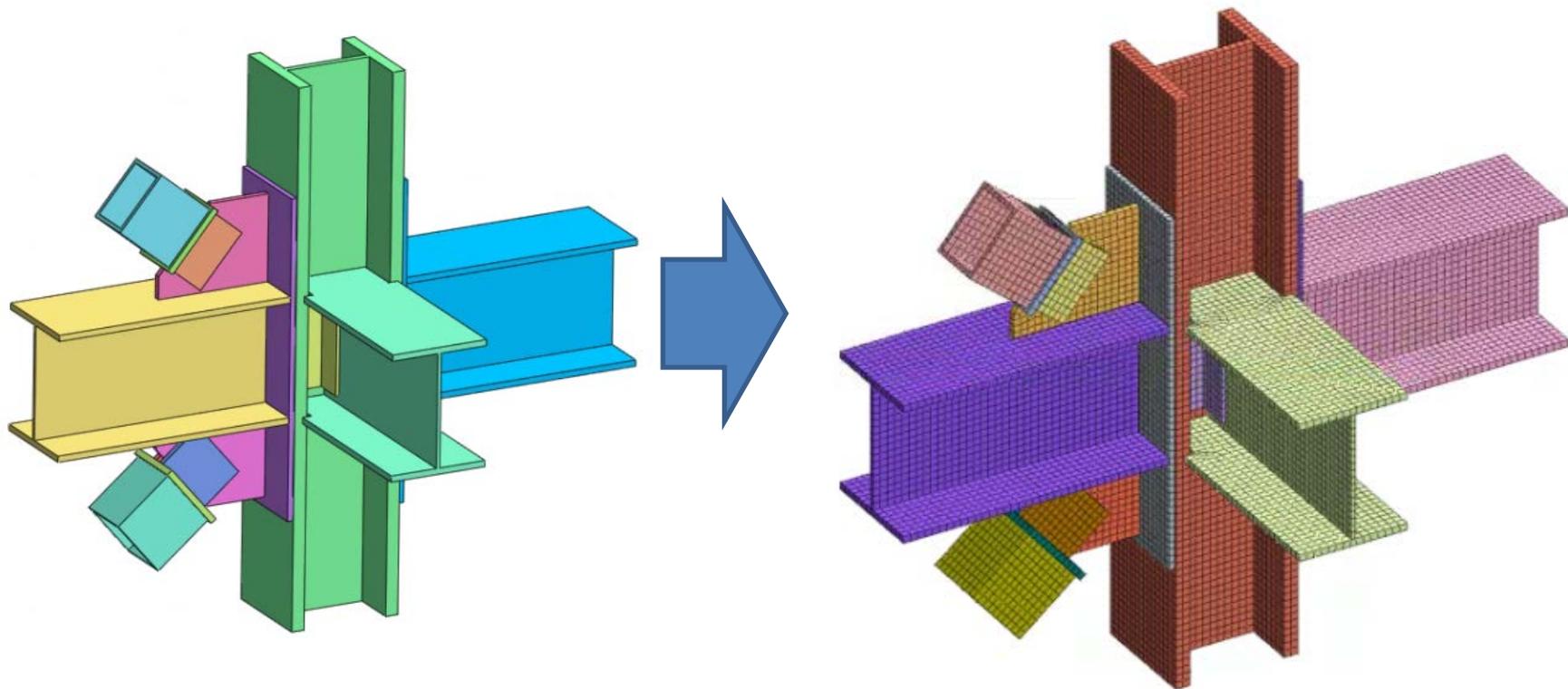


3D網格-混合網格



Hybrid Mesher(混合網格)/網格尺寸0.02 (m)/1階元素

參考相鄰特徵/不合併節點

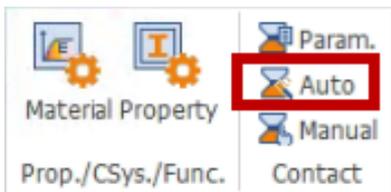


補充

操作視窗右下角顯示整體模型網格和節點數

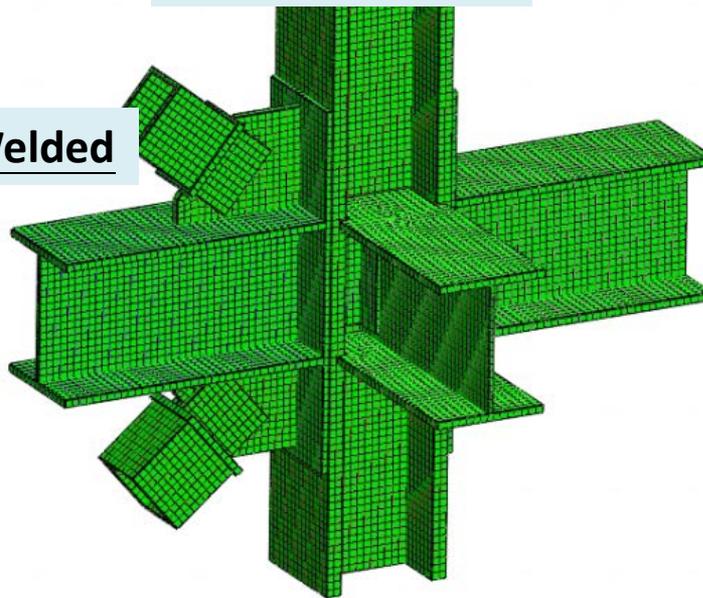


自動焊接接觸



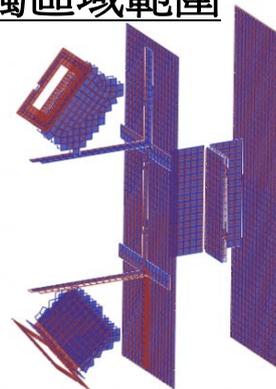
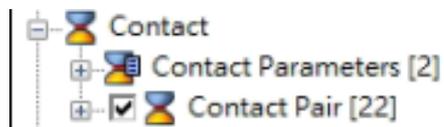
預設接觸類型:Welded

選取所有網格集

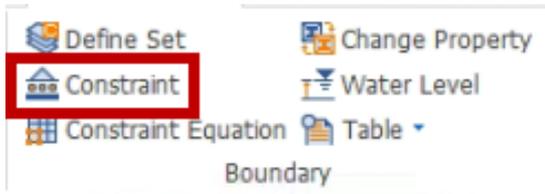


補充

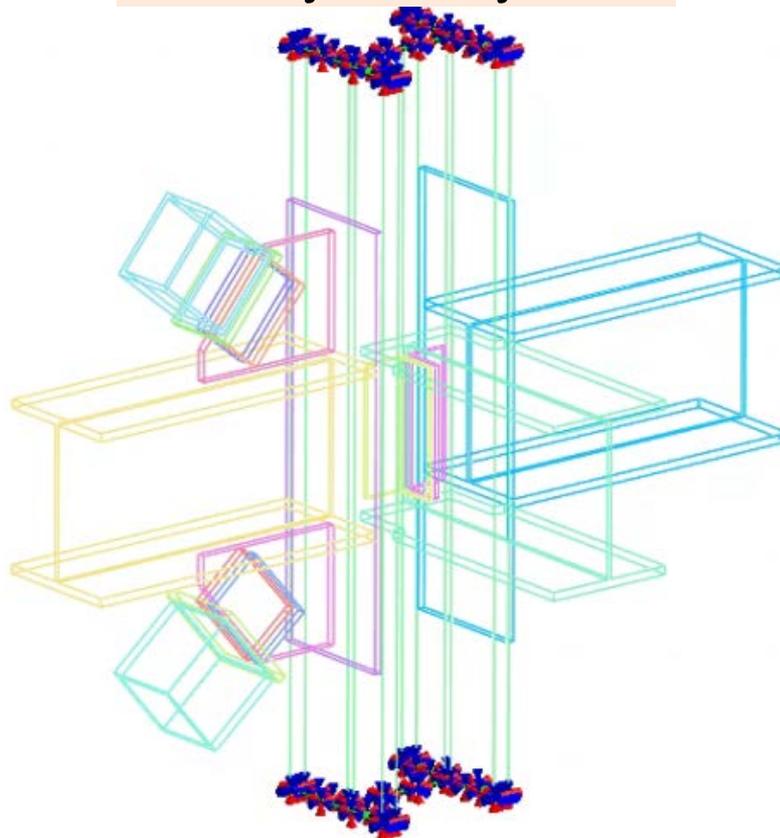
點選接觸對檢視接觸區域範圍



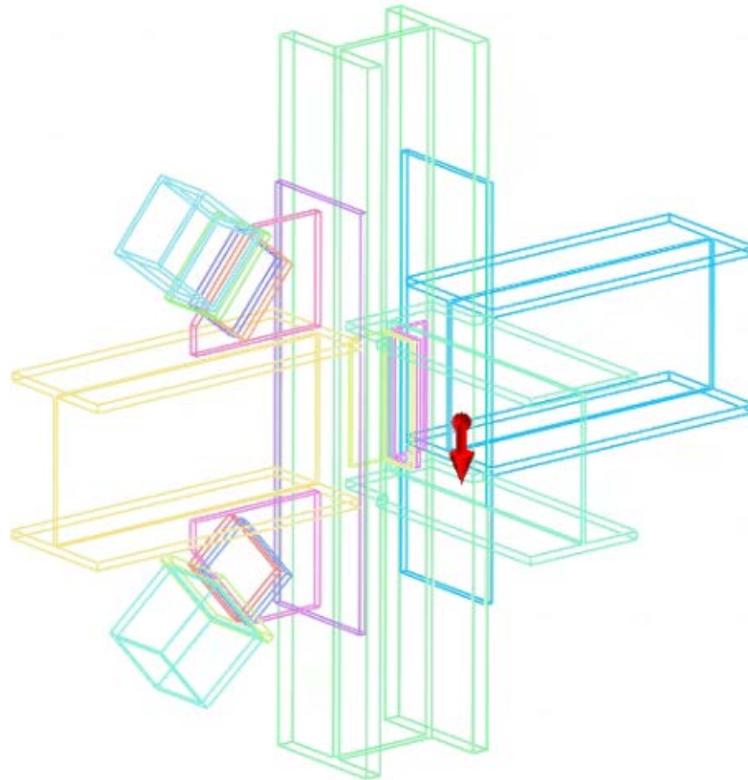
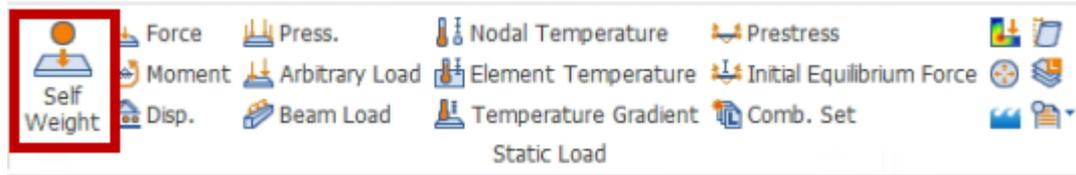
邊界



拘束立柱上下面特徵
 $T_x/T_y/T_z/R_x/R_y/R_z$



自重



載荷-1

(全局座標判斷方向)

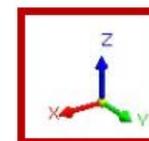
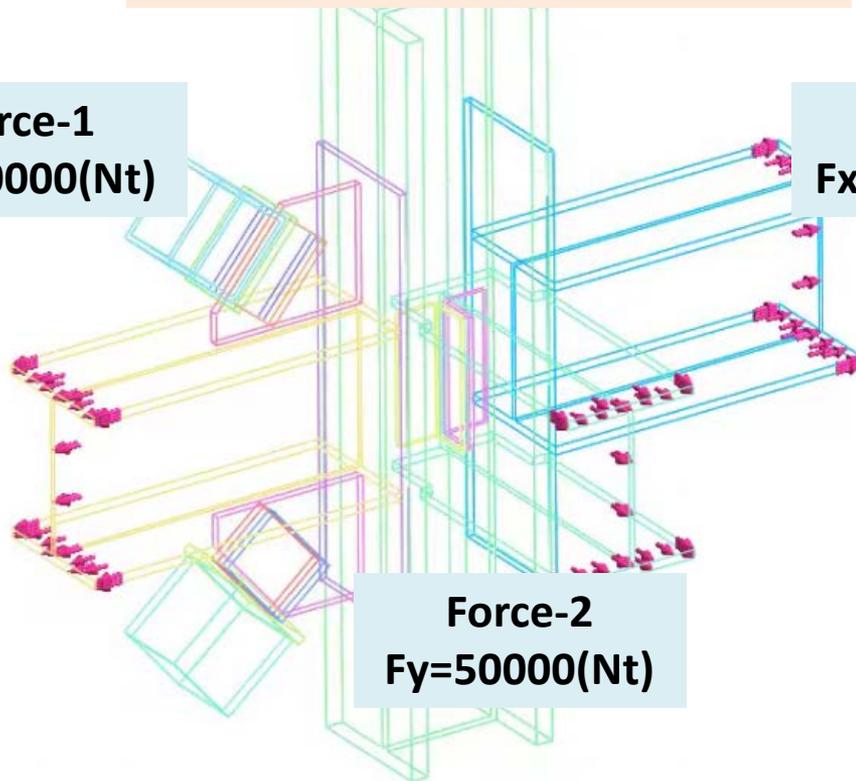


鋼構橫樑端施加50000(Nt)

Force-1
 $F_x=50000(\text{Nt})$

Force-3
 $F_x=-50000(\text{Nt})$

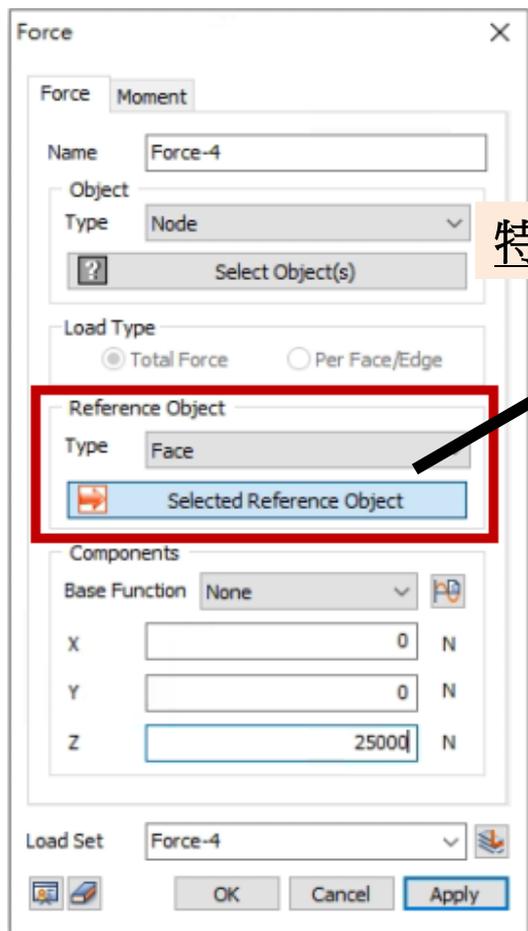
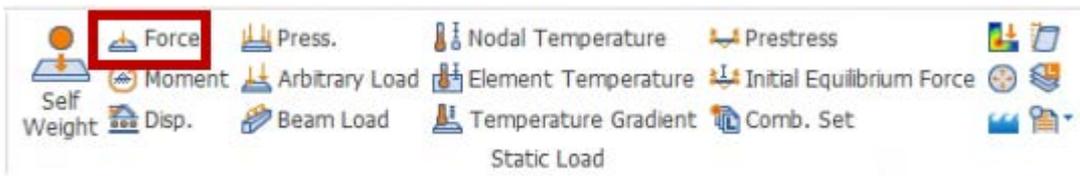
Force-2
 $F_y=50000(\text{Nt})$



全局座標

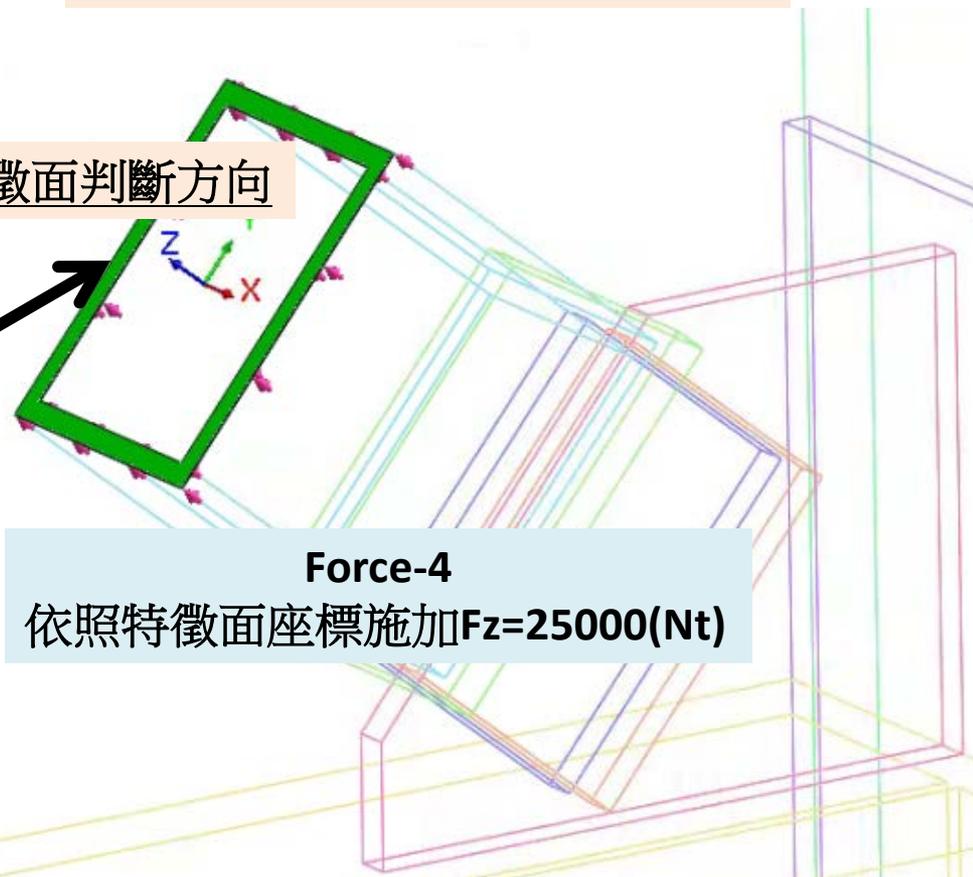
載荷-2

(特徵面判斷方向)



鋼構斜拉端施加25000(Nt)

特徵面判斷方向

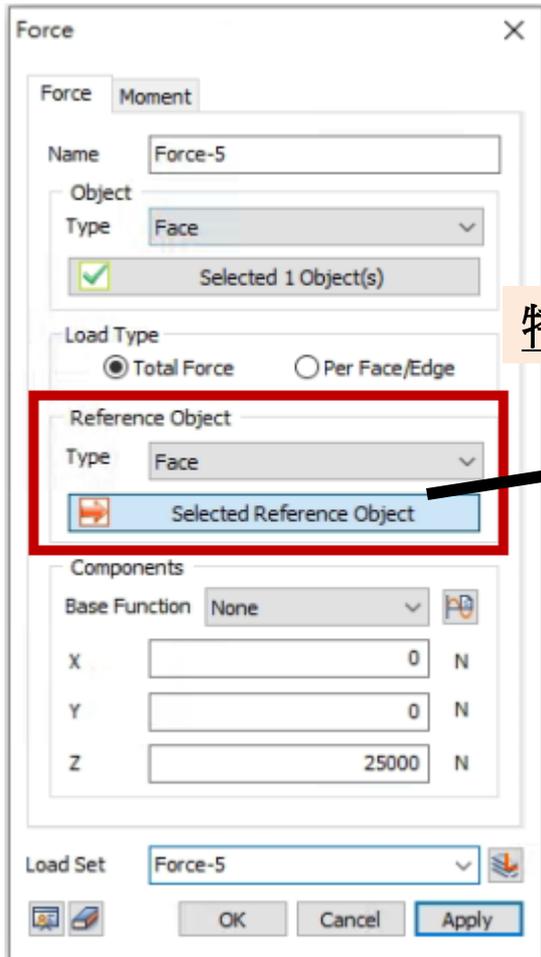
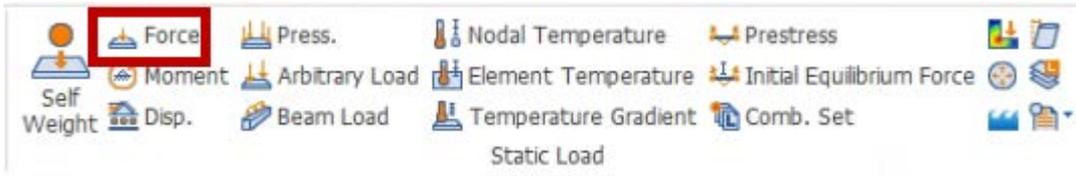


Force-4

依照特徵面座標施加 $F_z=25000(Nt)$

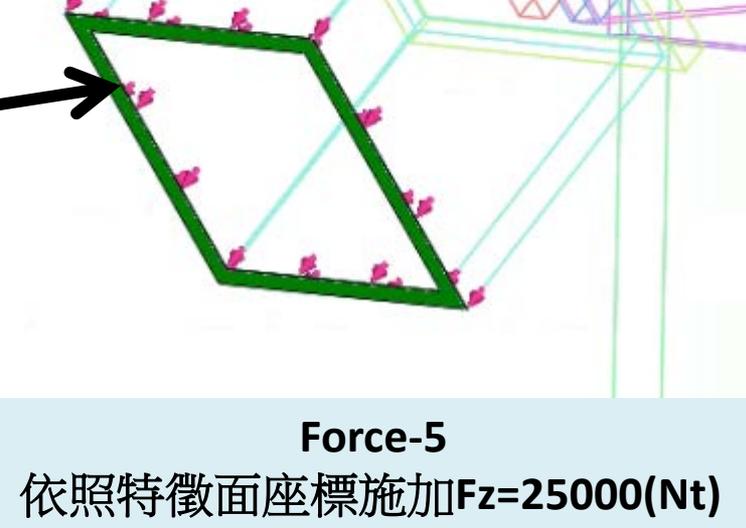
載荷-3

(特徵面判斷方向)



鋼構斜拉端施加25000(Nt)

特徵面判斷方向





線性分析-多載荷計算

自訂析名稱

Title: 多載荷計算

Solution Type: **Linear Static**

分析類型: Linear Static

考慮所有網格集/邊界集/載荷集/接觸對

Analysis Case Model

- All Sets
 - Mesh
 - Auto-Mesh(3D)
 - Auto-Mesh(3D)-1
 - Auto-Mesh(3D)-10
 - Auto-Mesh(3D)-11
 - Auto-Mesh(3D)-12
 - Auto-Mesh(3D)-13
 - Auto-Mesh(3D)-17
 - Auto-Mesh(3D)-2
 - Auto-Mesh(3D)-3
 - Auto-Mesh(3D)-4
 - Auto-Mesh(3D)-5
 - Auto-Mesh(3D)-6
 - Auto-Mesh(3D)-7
 - Auto-Mesh(3D)-8
 - Auto-Mesh(3D)-9
 - Default Mesh Set
 - Boundary Condition
 - Boundary Set-1
 - Static Load
- Active Sets
 - Mesh
 - Auto-Mesh(3D)
 - Auto-Mesh(3D)-1
 - Auto-Mesh(3D)-10
 - Auto-Mesh(3D)-11
 - Auto-Mesh(3D)-12
 - Auto-Mesh(3D)-13
 - Auto-Mesh(3D)-17
 - Auto-Mesh(3D)-2
 - Auto-Mesh(3D)-3
 - Auto-Mesh(3D)-4
 - Auto-Mesh(3D)-5
 - Auto-Mesh(3D)-6
 - Auto-Mesh(3D)-7
 - Auto-Mesh(3D)-8
 - Auto-Mesh(3D)-9
 - Default Mesh Set
 - Boundary Condition
 - Boundary Set-1
 - Static Load

Output Control

輸出控制

Output Control

Output Type

Write Results of All Active Mesh Sets

Nodal Results

- Displacement
- Applied Load
- Reaction Force
- Grid Point Force
- Contact

Element Results

- Force
- Stress
- Strain
- Status
- Multi-layered Grid

Output Option

Binary Binary and Text

Element Output Location

- Element Corner Results
- Shell Mid-Plane Results

Number of Beam Output Segments: 4

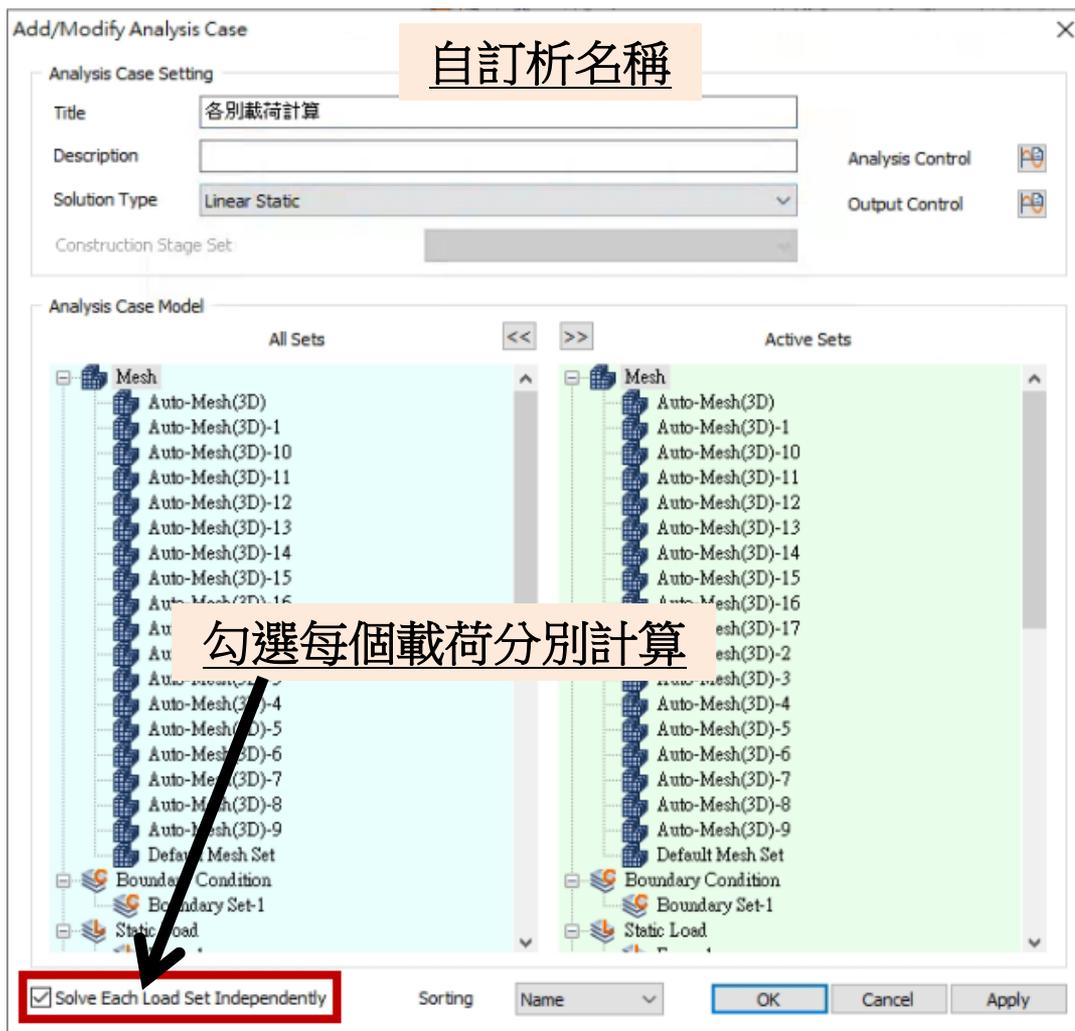
OK Cancel

接觸結果輸出



線性分析-各別載荷計算

求解定義與多載荷計算相同



執行分析

執行分析

FEA NX Solver

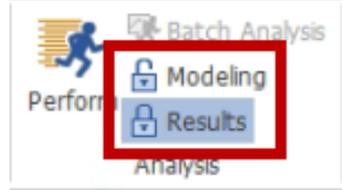
Name	Type	Description
<input checked="" type="checkbox"/> 多載荷計算	near Static	
<input checked="" type="checkbox"/> 各別載荷計算	near Static	

勾選執行分析項目

OK Cancel

W: 2.32205, -0.260848 X: -0.92599, -0.92599 Y: -0.15499, -0.707 Z: -0.83499, -0.83499 6 [18] M [29135] E [15878]

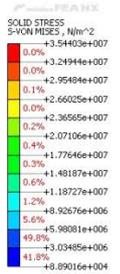
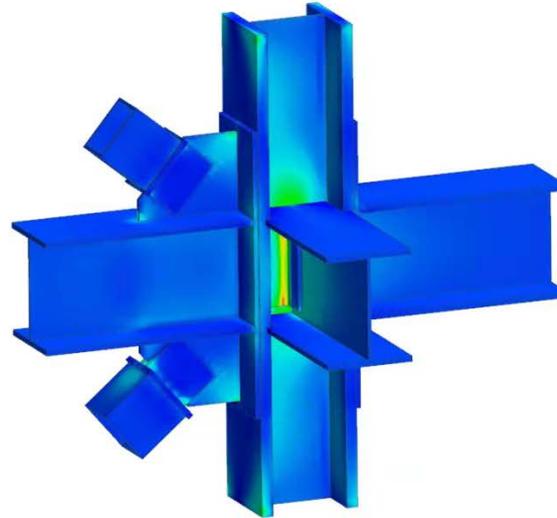
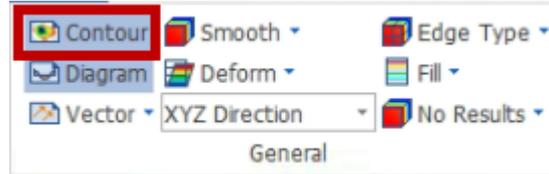
前後處理切換



各別載荷結果

分析結果-1

(輪廓圖)

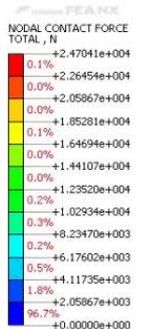
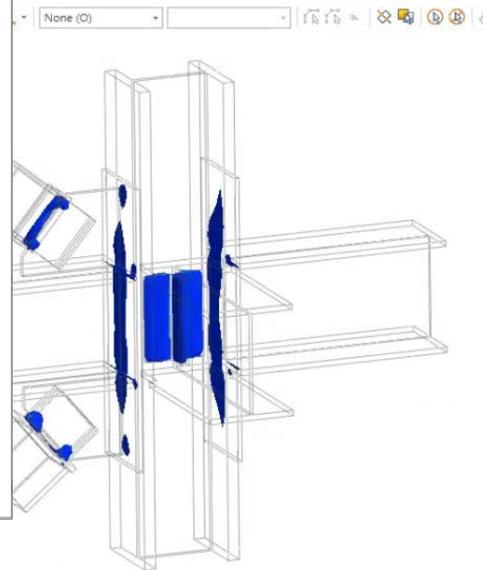
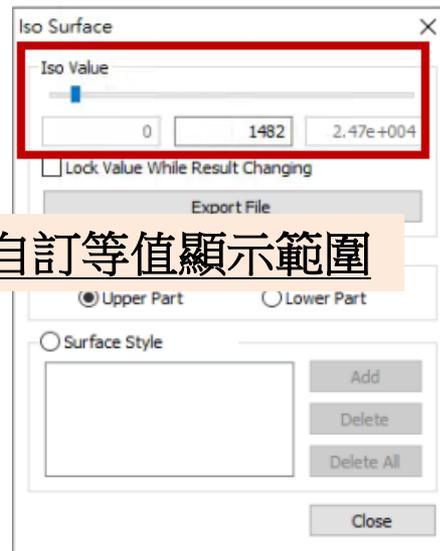


[DATA] 多載荷計算, Linear Static, [UNIT] N, m, [Output Csys] Default

分析結果-2

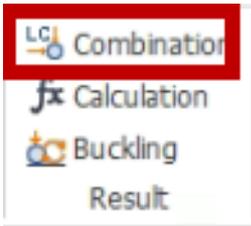
(等值圖)

等值圖



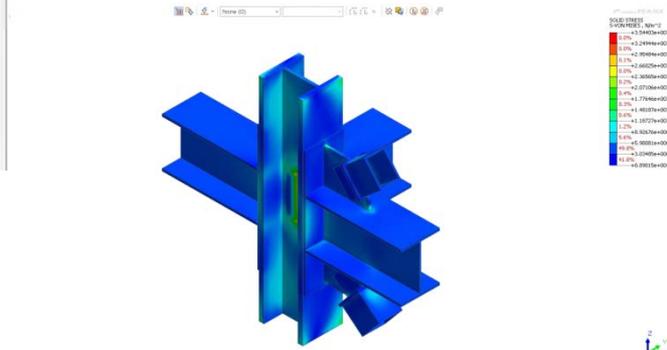
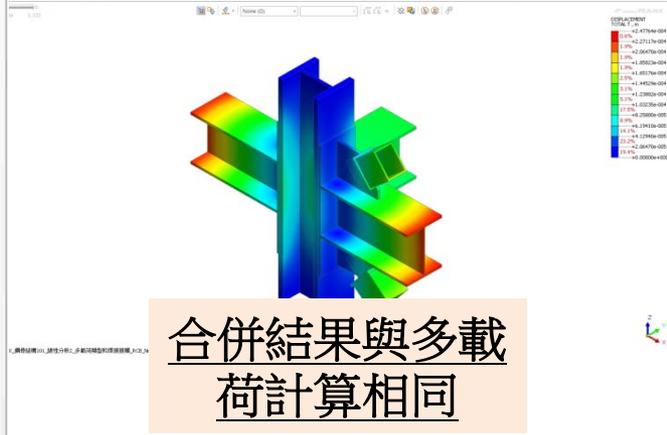
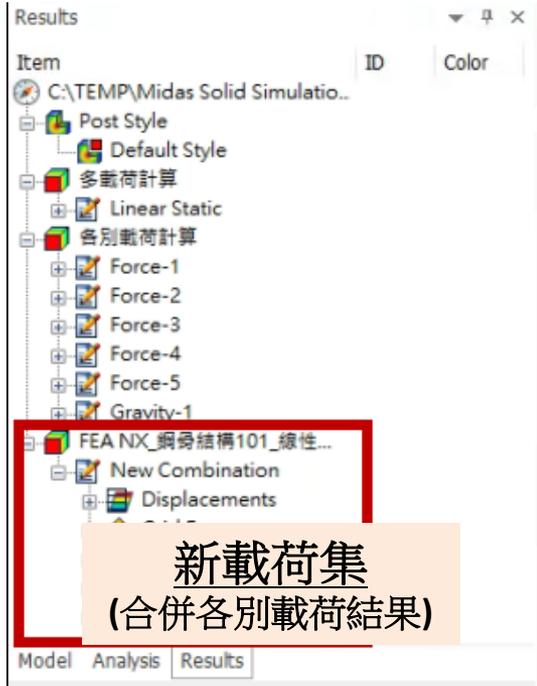
[DATA] 多載荷計算, Linear Static, [UNIT] N, m, [Output CSys] Default





分析結果-3

(合併結果)



[DATA] FEA NX_鋼骨結構101_線性分析; 多載荷結果與合併結果;_FCI-New-Combination; New-Combination; [UNIT] N, m; [Output Class] Default

FEANX_標準教學系列

鋼骨結構-線性分析
線性挫曲分析

台灣邁達斯

挫曲

挫曲(Buckling)

是指細長桿件受到壓力時，發生彎曲變形的一種現象，不穩定造成的結構失效稱為屈曲失效，理想壓桿不穩定後，由原來的直線平衡狀態變為彎曲平衡狀態。

數學家萊昂哈德·歐拉提出了細長理想柱在不挫曲的情形下，可以承受的最大軸向壓縮力。

$$F = \frac{\pi^2 EI}{(KL)^2}$$

F:最大臨界力

E:彈性模數

I:面積慣性矩

L:柱的未支撐長度

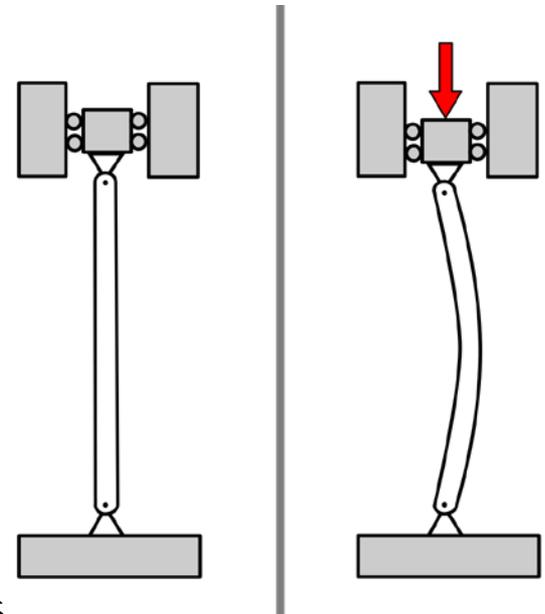
K:視柱子兩側的支撐條件而定

K=1,二端都用插銷連接(可以旋轉)

K=0.5,若二端都固定(無法旋轉)

K≈ 0.699,一端固定一端用插銷連接

K=2,一端固定，另一端可以自由移動



Reference

<https://zh.wikipedia.org/zh-tw/挫曲>

分析說明

材料

$$E = 70\text{GPa}$$

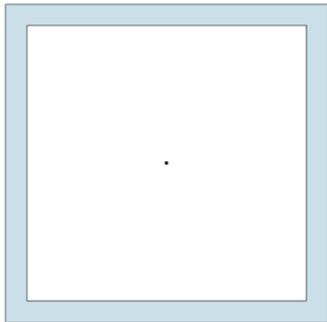
$$\gamma = 0.33$$

$$\rho = 7.850\text{kg/m}^3$$

截面0.15(m) x 0.15(m)

厚度(t):0.01(m)

長度(L):3 (m)



施加160,000(N)
(未固定)

F最大臨界力:352.86(KN)

$$F = \frac{\pi^2 EI}{(KL)^2}$$

E:70(GPa)

L:3(m)

K=2

I: $\frac{1}{12}((0.15 \times 0.15^3) - (0.13 \times 0.13^3))$

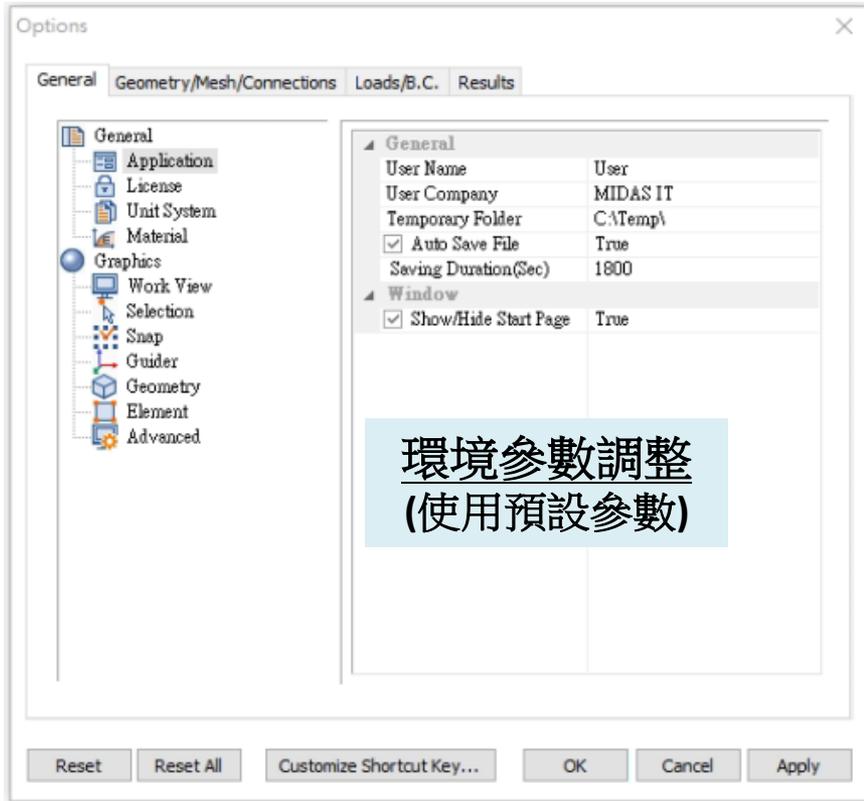
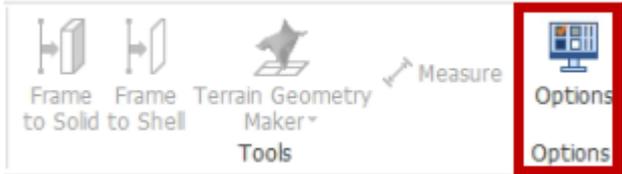


線性挫曲分析
計算最大臨界力

拘束條件

底部面特徵Tx/Ty/Tz/Rx/Ry/Rz

環境

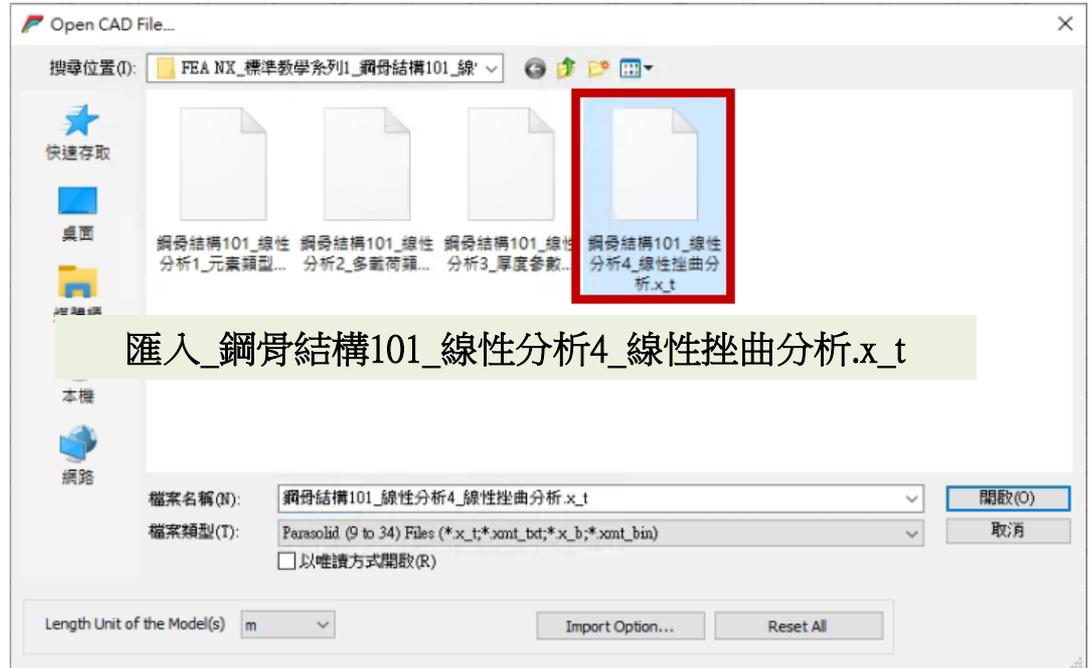
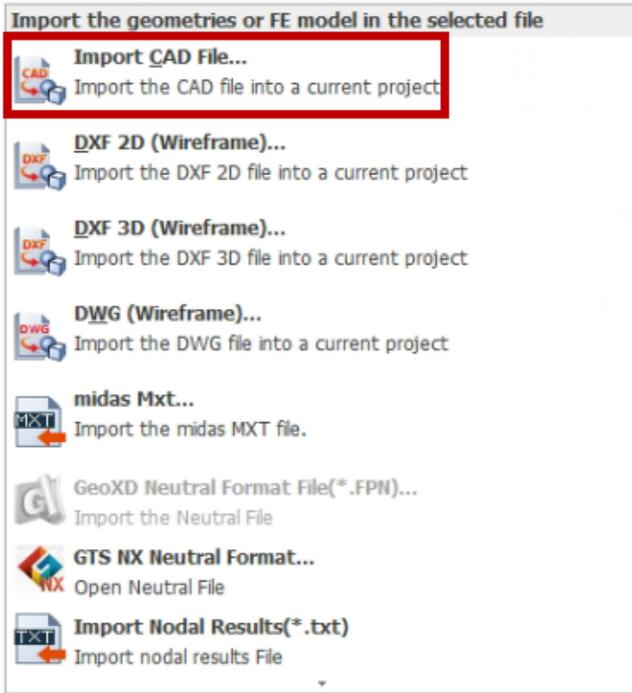
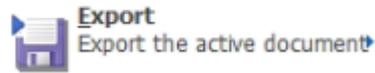


新文件



單位使用N/m/J/sec

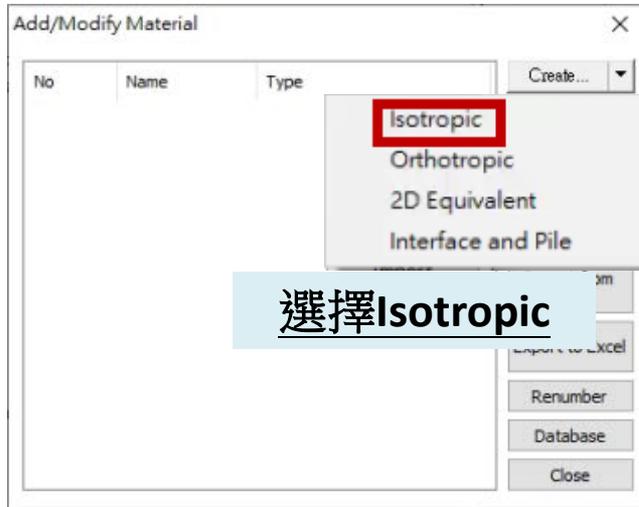
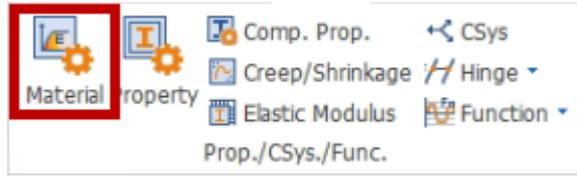
匯入模型



匯入_鋼骨結構101_線性分析4_線性挫曲分析.x_t

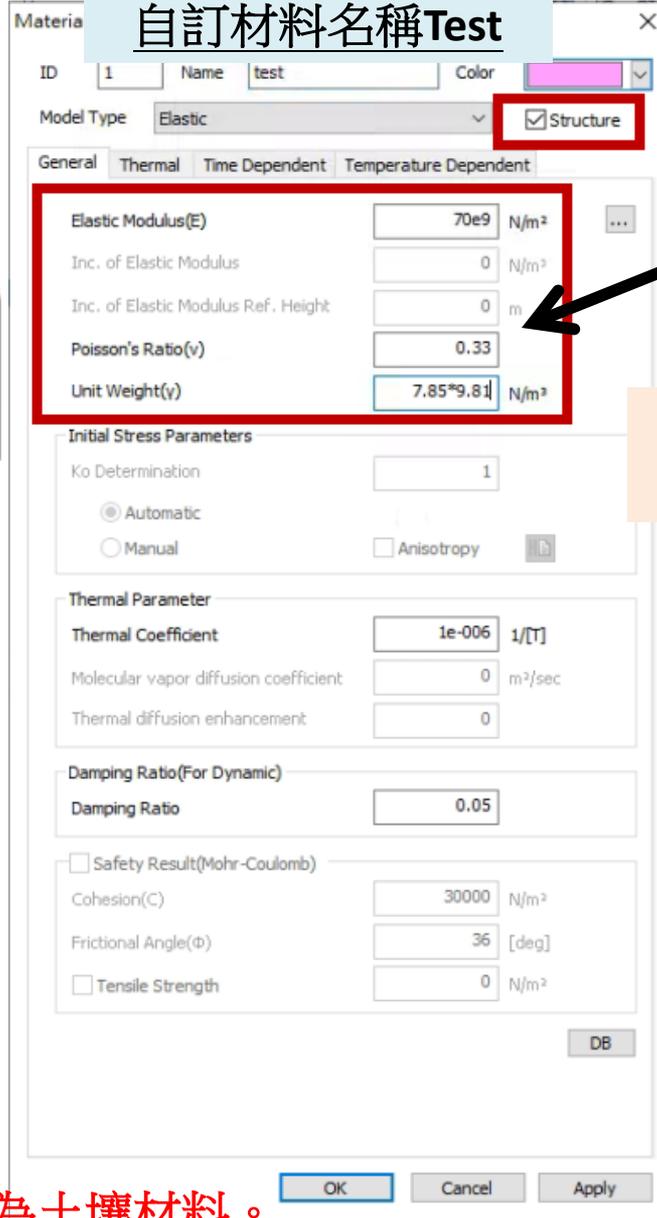
鋼構3D模型

材料



選擇Isotropic

自訂材料名稱Test



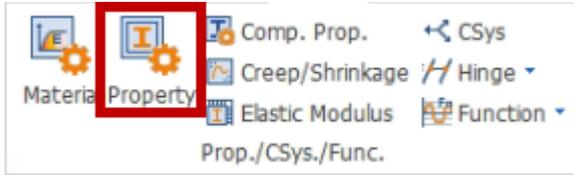
勾選Structure

$$E = 70GPa$$
$$\gamma = 0.33$$
$$\rho = 7.850kg / m^3$$

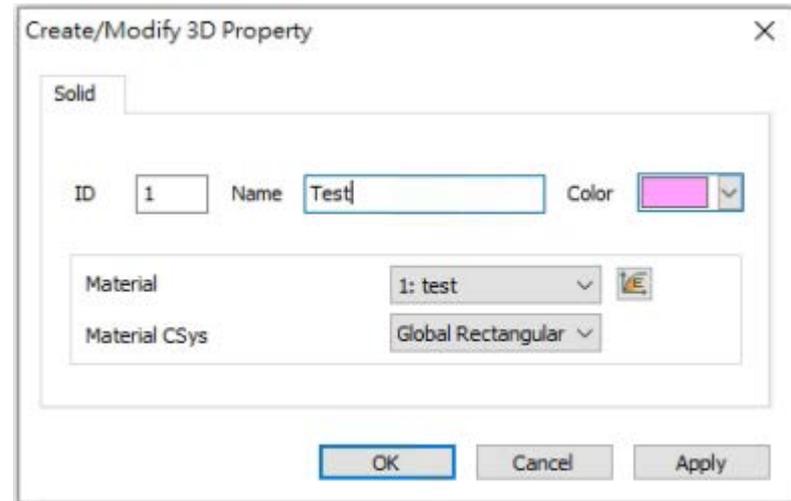
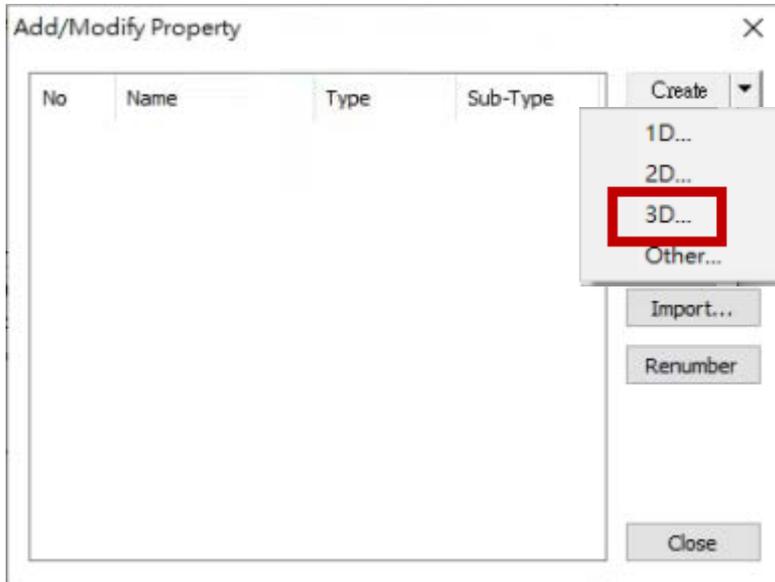
對照輸入材料數據
(密度單位換算)

註:未勾選Structure則視為土壤材料。

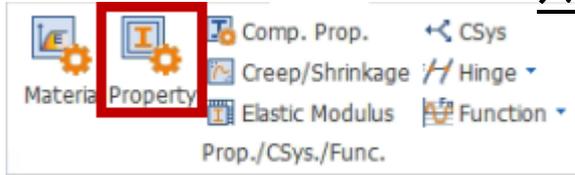
屬性



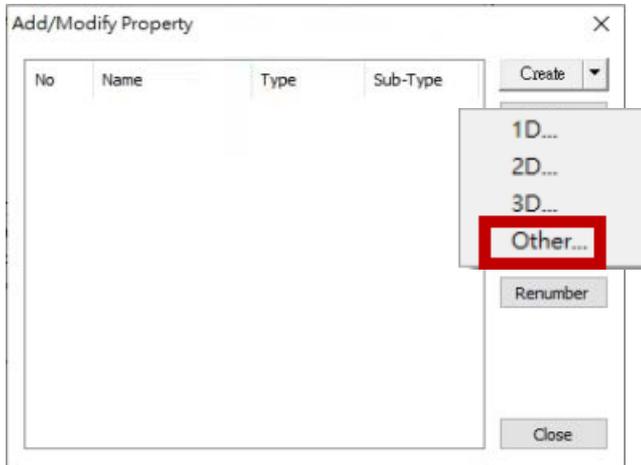
新增Test Property



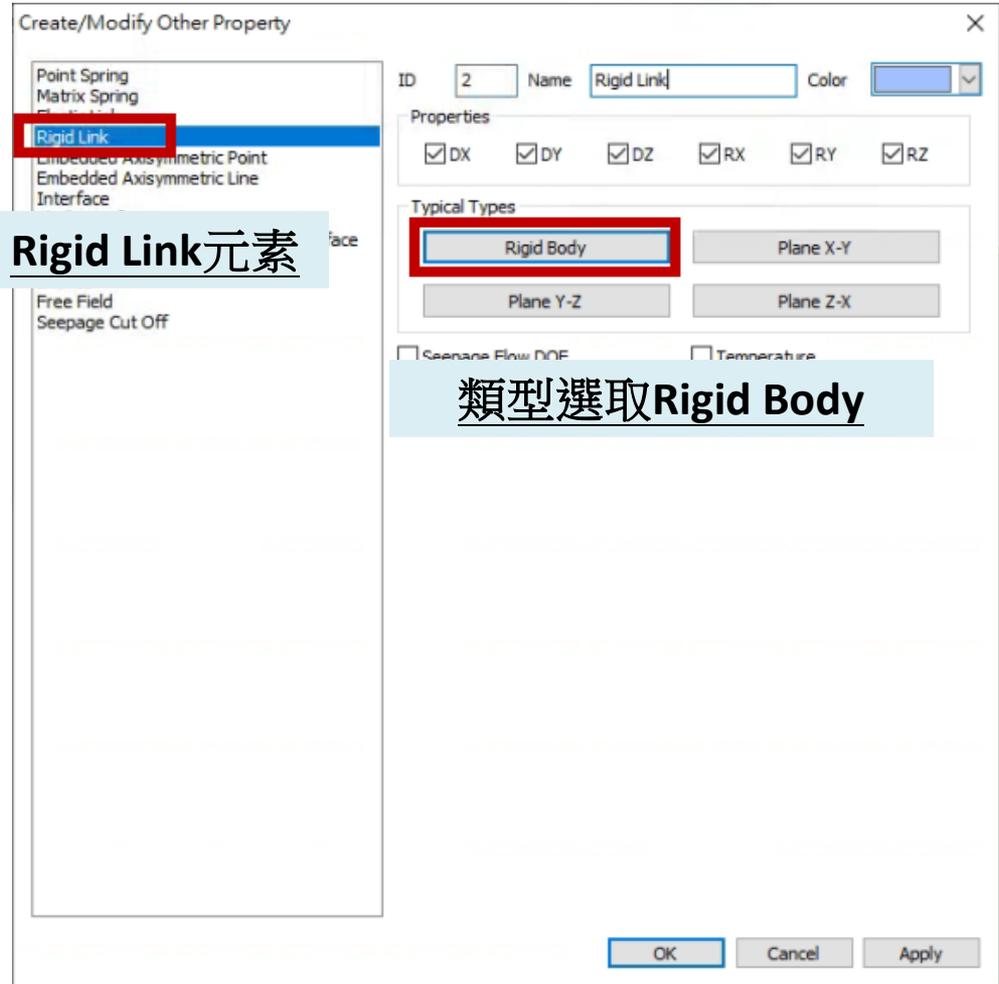
屬性-Rigid Link



新增Rigid Link Property



Rigid Link元素

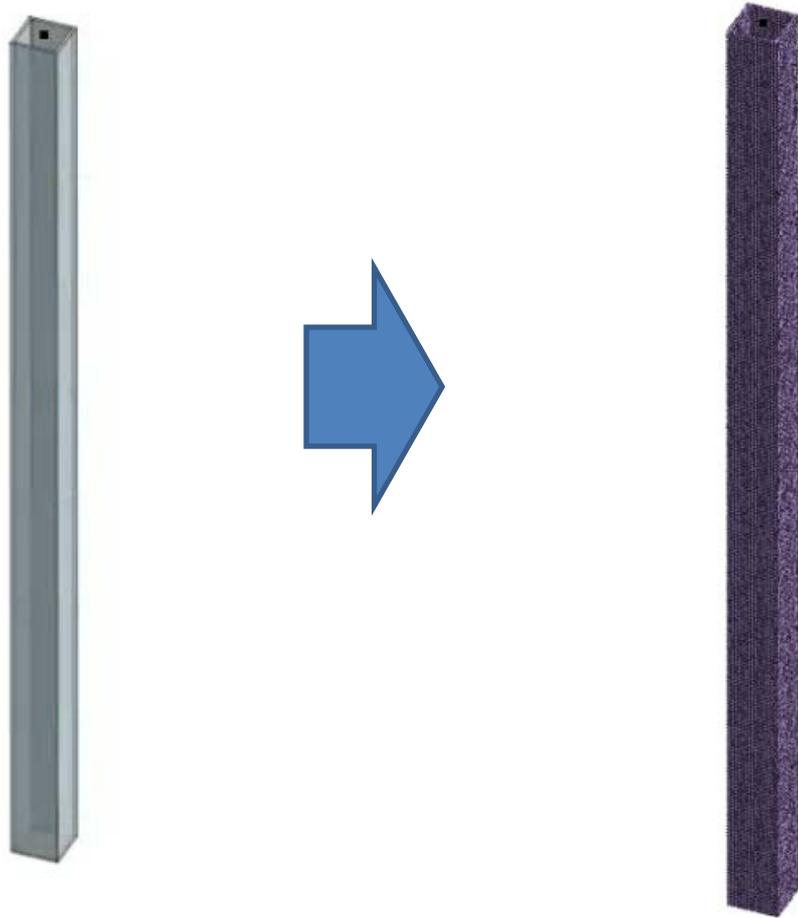


類型選取Rigid Body

3D網格-四面體網格

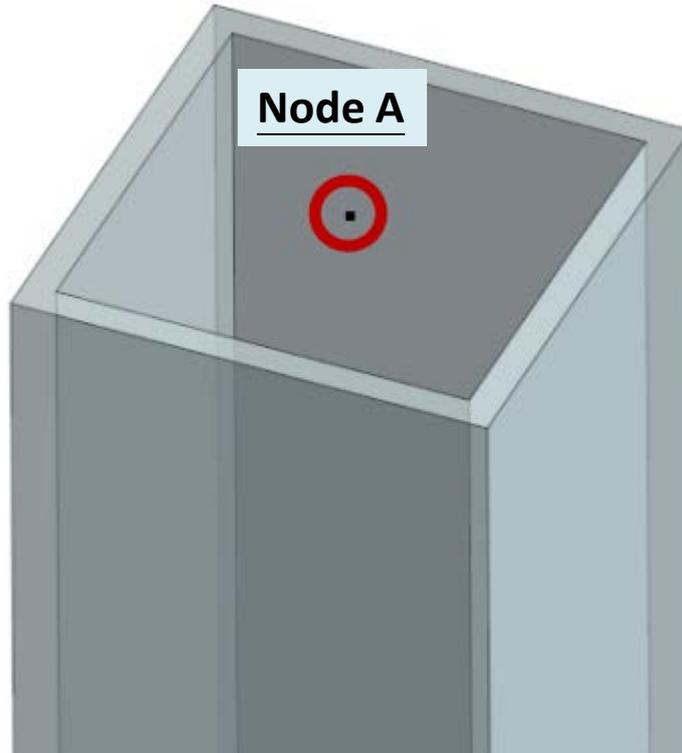
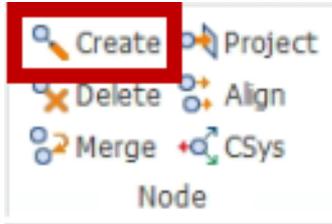


Tetra Mesher(四面體網格)/網格尺寸0.01 (m)/2階元素

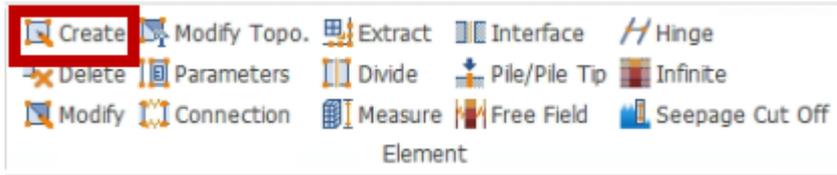


主控節點建立

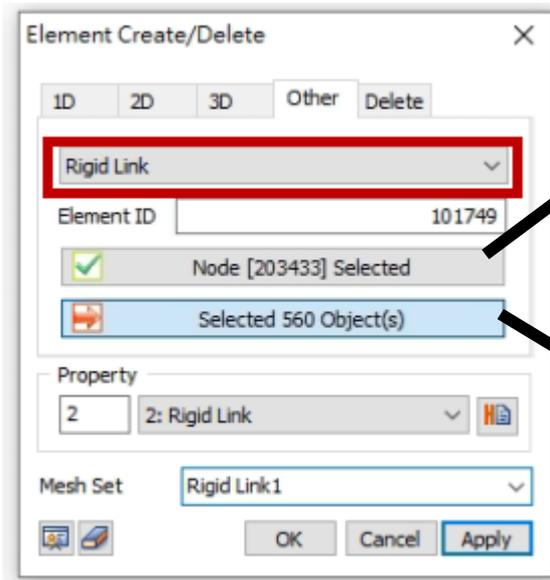
(剛性連接主控點)



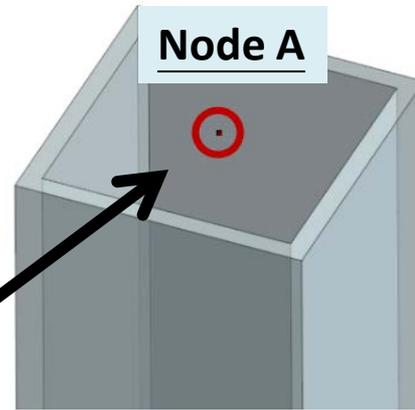
剛性連接



元素類型選取Rigid Link

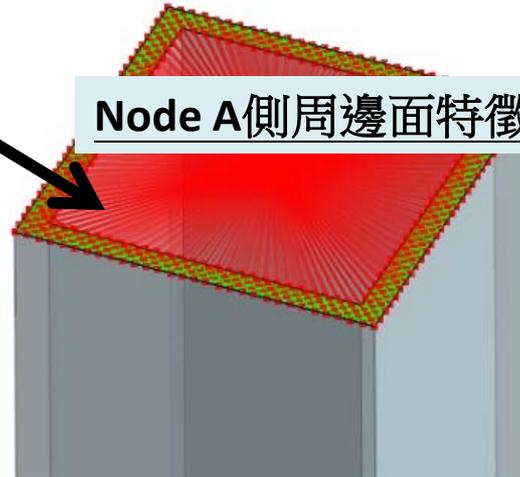


自訂網格集Rigid Link1

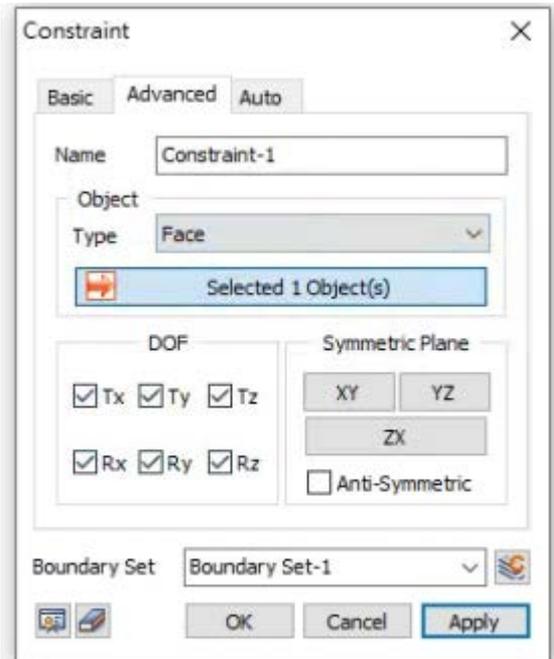
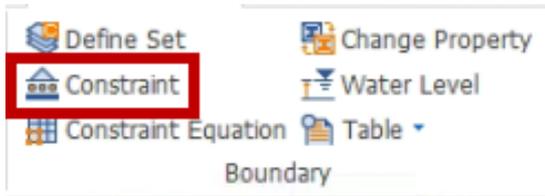


切換使用面特徵選取節點

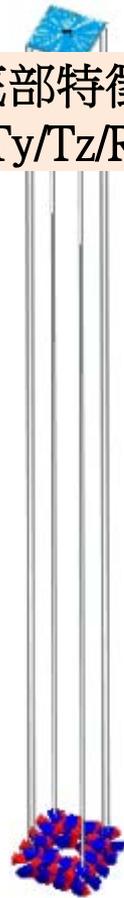
Node A側周邊面特徵節點



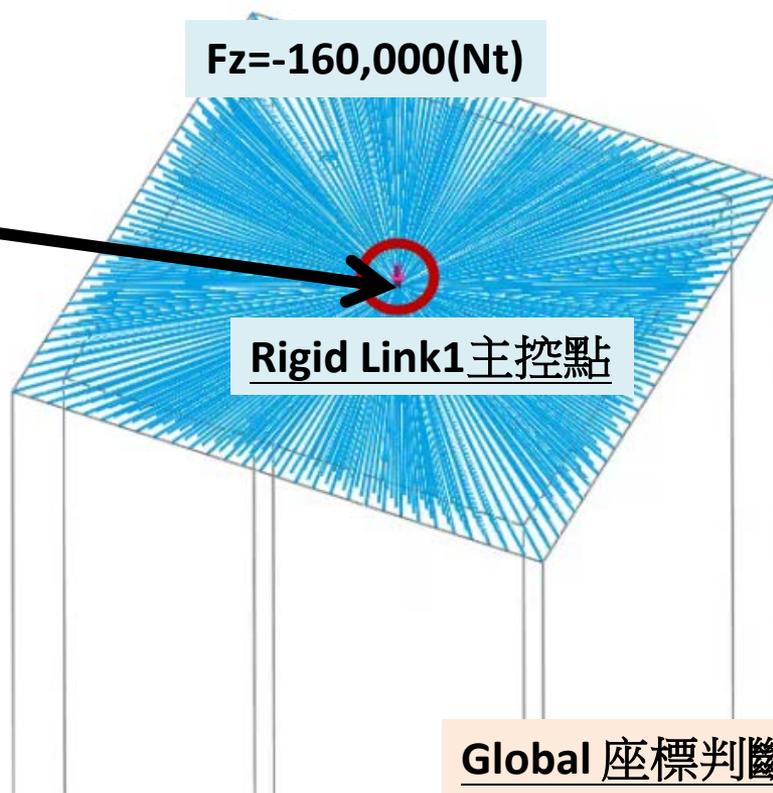
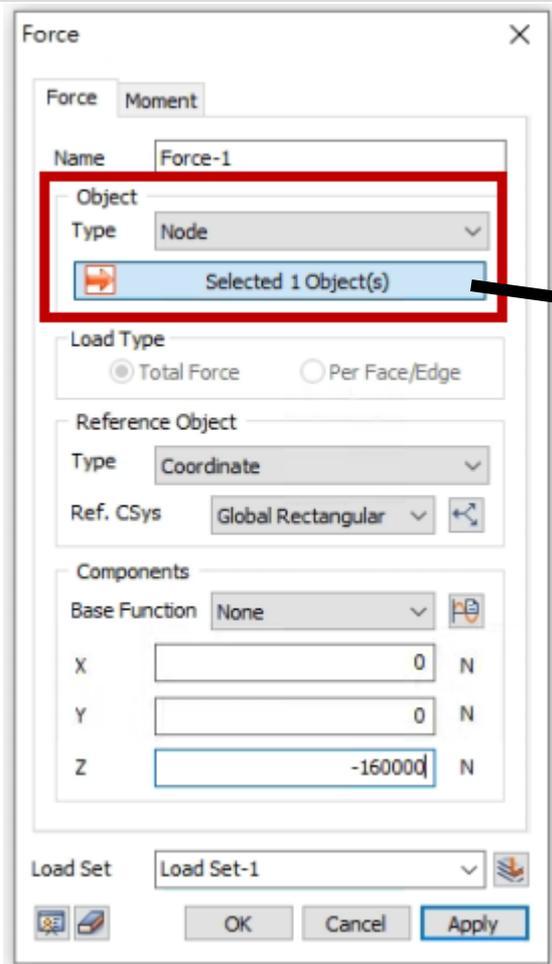
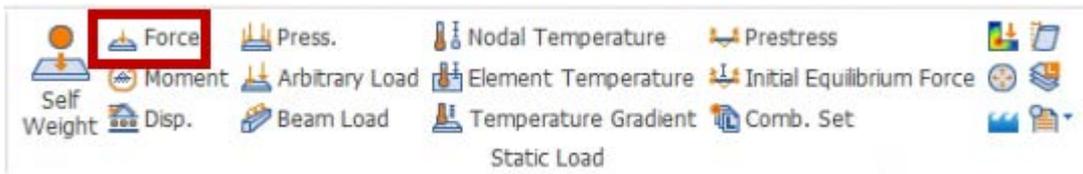
邊界



底部特徵拘束
Tx/Ty/Tz/Rx/Ry/Rz



載荷



$F_z = -160,000(Nt)$

Rigid Link1 主控點

Global 座標判斷方向

Setting
Parametric Analysis
General
Analysis Case

線性挫曲分析-1

自訂析名稱

Analysis Case Setting

Title: 線性挫曲分析

Description:

Solution Type: Linear Buckling

Analysis Control

Output Control

Construction Stage

分析類型: Linear Buckling

Analysis Case Mod

All Sets

- Mesh
 - Auto-Mesh(3D)
 - Default Mesh Set
 - Node A
 - Rigid Link1
- Boundary Condition
 - Boundary Set-1
- Static Load

Active Sets

- Mesh
 - Auto-Mesh(3D)
 - Default Mesh Set
 - Node A
 - Rigid Link1
- Boundary Condition
 - Boundary Set-1
- Static Load

考慮所有網格集/邊界集/載荷集/接觸對

Solve Each Load Set Independently

Sorting: Name

OK Cancel Apply

線性挫曲分析-2

分析控制

Analysis Control

General

Initial Temperature

Initial Temperature By Value [T]

Load Set Control

Load Set	Factor	Load Type
1:Load Set-1	1.0000	Variable

Eigenvectors

Number of Modes

Frequency Range of Interest

Lowest Highest

Unit: [Cycle]/ sec

Sturm Sequence Check

OK Cancel

輸出10組結果

輸出控制

Output Control

Output Type Output Option

Write Results of All Active Mesh Sets

Nodal Results

Displacement

Applied Load

Reaction Force

Grid Point Force

Contact

Element Results

Force

Stress

Strain

Output Option

Binary Binary and Text

Element Output Location

Element Corner Results

Shell Mid-Plane Results

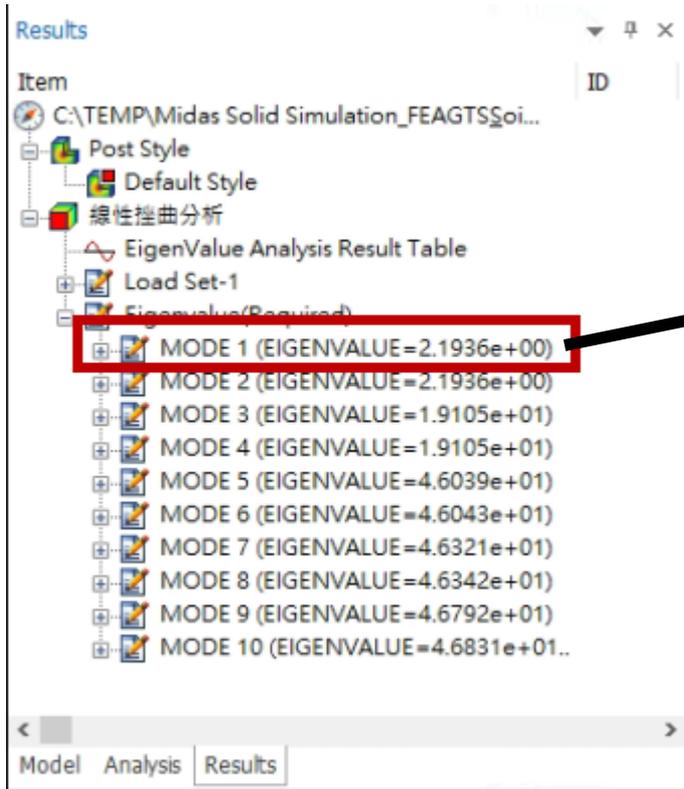
Number of Beam Output Segments

OK Cancel

自訂輸出項目

挫曲計算

線性挫曲分析



n=1

$$160(\text{KN}) \times 2.1936 = 350.976(\text{KN})$$

F最大臨界力:352.86(KN)

$$F = \frac{\pi^2 EI}{(KL)^2}$$

E:70(GPa)

L:3(m)

K=2

I: $\frac{1}{12}((0.15 \times 0.15^3) - (0.13 \times 0.13^3))$

Reference

NFX Analysis Reference

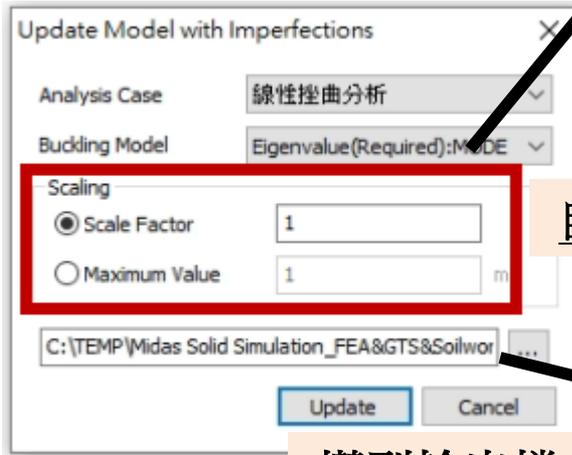


挫曲變形模型



選取結果

```
Eigenvalue(Required):MODE 1 (EIGENVALUE=2.1936e+00)  
Eigenvalue(Required):MODE 2 (EIGENVALUE=2.1936e+00)  
Eigenvalue(Required):MODE 3 (EIGENVALUE=1.9105e+01)  
Eigenvalue(Required):MODE 4 (EIGENVALUE=1.9105e+01)  
Eigenvalue(Required):MODE 5 (EIGENVALUE=4.6039e+01)  
Eigenvalue(Required):MODE 6 (EIGENVALUE=4.6043e+01)  
Eigenvalue(Required):MODE 7 (EIGENVALUE=4.6321e+01)  
Eigenvalue(Required):MODE 8 (EIGENVALUE=4.6342e+01)  
Eigenvalue(Required):MODE 9 (EIGENVALUE=4.6792e+01)  
Eigenvalue(Required):MODE 10 (EIGENVALUE=4.6831e+01)
```



自訂變形比例

模型輸出檔名



FEA NX_標準教學系列

鋼骨結構-非線性分析
挫曲計算

台灣邁達斯

挫曲

挫曲(Buckling)

是指細長桿件受到壓力時，發生彎曲變形的一種現象，不穩定造成的結構失效稱為屈曲失效，理想壓桿不穩定後，由原來的直線平衡狀態變為彎曲平衡狀態。

數學家萊昂哈德·歐拉提出了細長理想柱在不挫曲的情形下，可以承受的最大軸向壓縮力。

$$F = \frac{\pi^2 EI}{(KL)^2}$$

F:最大臨界力

E:彈性模數

I:面積慣性矩

L:柱的未支撐長度

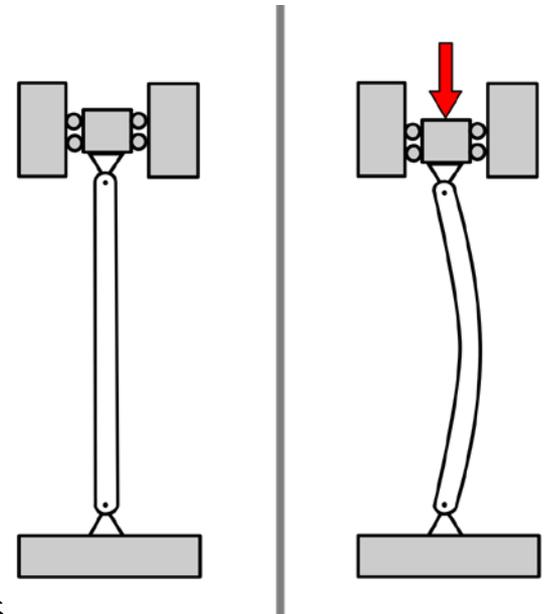
K:視柱子兩側的支撐條件而定

K=1,二端都用插銷連接(可以旋轉)

K=0.5,若二端都固定(無法旋轉)

K≈ 0.699,一端固定一端用插銷連接

K=2,一端固定，另一端可以自由移動



Reference

<https://zh.wikipedia.org/zh-tw/挫曲>

後挫曲

範例探討內容

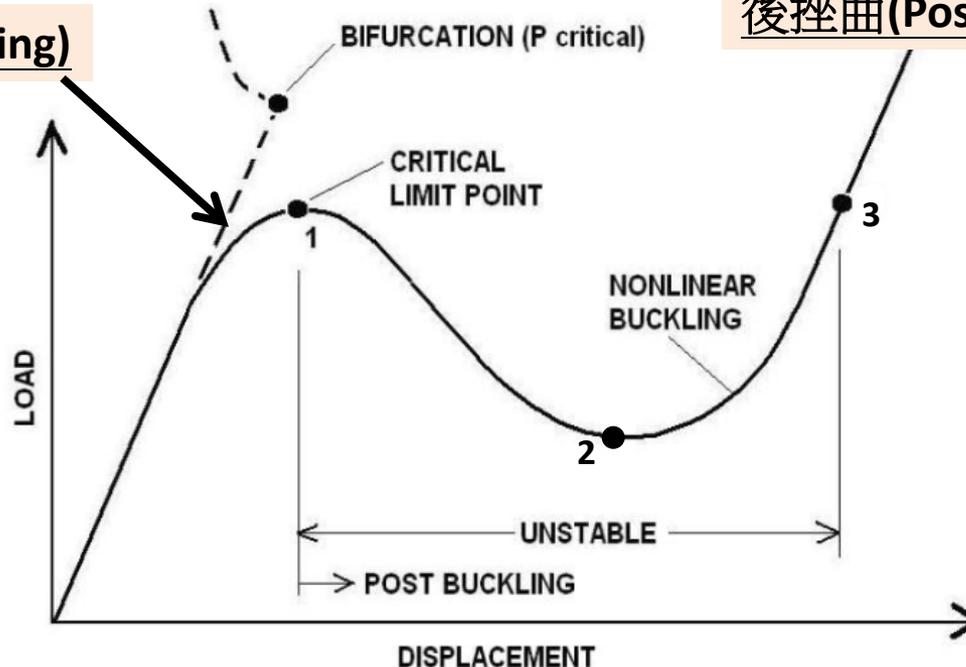
後挫曲(非線性挫曲)分析

載荷較小時分析結果與線性結果一致。當載荷逐步增大，變形逐步增加，結構響應呈現出非線性，力位移曲線開始偏離線性結果，在Point 1點承受外部載荷達到最大。

之後變形繼續變大但是結構能承受的外部載荷變小，結構發生挫曲，變形繼續變大，但是結構承受的載荷繼續變小。達到Point 2位移增加，結構承受的載荷開始增加，Point 3達到之前載荷的極限點。從Point 1到Point 3的過程稱為疾速跳過(snap through)。

挫曲(Buckling)

後挫曲(Post Buckling)



Reference

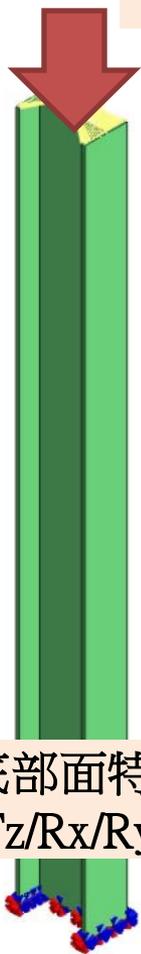
<https://www.linkedin.com/pulse/buckling-post-buckling-abraham-imam-muttaqin>

分析說明

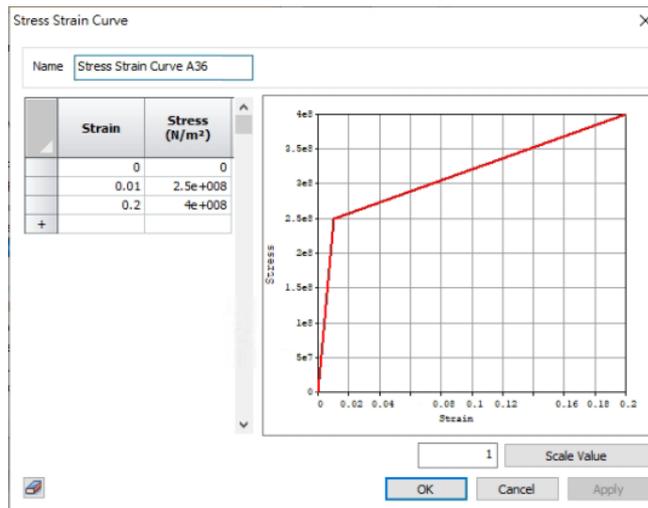
計算立柱下壓過程位移過程所需載荷!!

材料Steel/ASTM A36

下壓1(m)



立柱底部面特徵
Tx/Ty/Tz/Rx/Ry/Rz



ASTM A36 Steel, plate

Categories: Metal, Ferrous Metal, ASTM Steel, Carbon Steel, Low Carbon Steel

Material Notes: Steel for general structural purposes including bridges and buildings

Minimum Cu content when copper steel is specified.

Tests performed in transverse direction for plates wider than 590 mm.

Key Words: UNS K02600

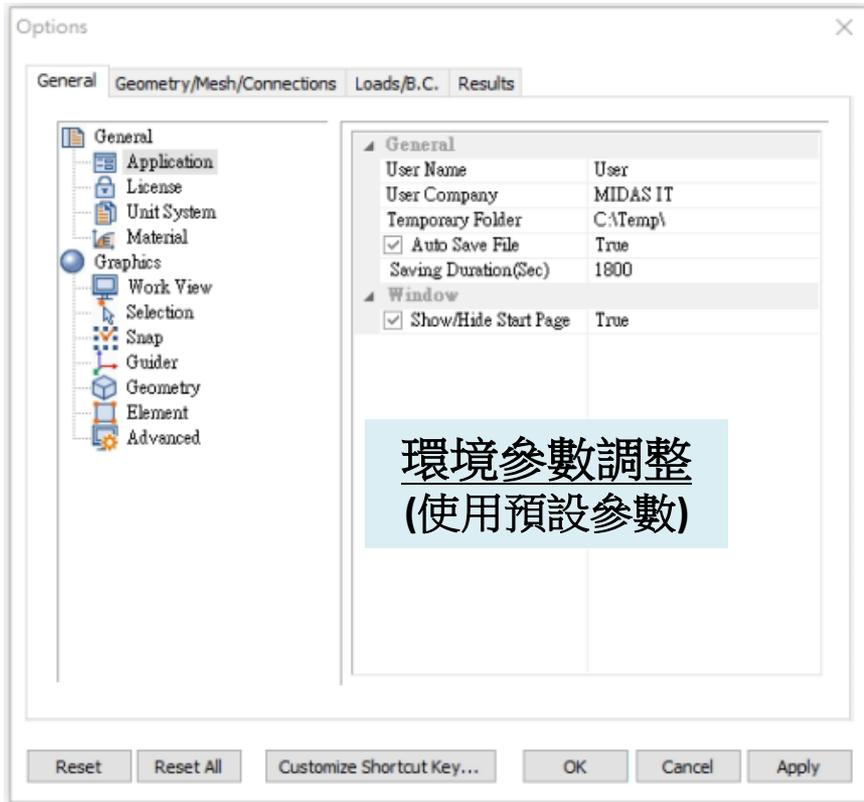
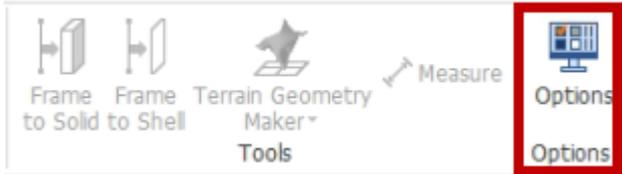
Vendors: No vendors are listed for this material. Please [click here](#) if you are a supplier and would like information on how to add your listing to this material.

[Print friendly version](#) [Download as PDF](#) [Download to Excel \(requires Excel and Windows\)](#)

[Export data to your CAD/FEA program](#)

Physical Properties	Metric	English
Density	7.80 g/cc	0.282 lb/in³
Mechanical Properties	Metric	English
Tensile Strength, Ultimate	400 - 550 MPa	58000 - 79000 psi
Tensile Strength, Yield	250 MPa	36300 psi
Elongation at Break	20 %	20 %
	23 %	23 %
Modulus of Elasticity	200 GPa	29000 ksi
Bulk Modulus	160 GPa	23200 ksi
Poisson's Ratio	0.26	0.26
Shear Modulus	79.3 GPa	11500 ksi
Component Elements Properties	Metric	English
Carbon, C	0.25 - 0.29 %	0.25 - 0.29 %
Copper, Cu	0.20 %	0.20 %
Iron, Fe	98 %	98 %
Manganese, Mn	1.03 %	1.03 %
Phosphorus, P	<= 0.040 %	<= 0.040 %
Silicon, Si	0.28 %	0.28 %
Sulfur, S	<= 0.050 %	<= 0.050 %

環境



新文件



單位使用N/m/J/sec

匯入模型

Import
Import the selected file

Export
Export the active document

Import the geometries or FE model in the selected file

Import CAD File...
Import the CAD file into a current project

DXF 2D (Wireframe)...
Import the DXF 2D file into a current project

DXF 3D (Wireframe)...
Import the DXF 3D file into a current project

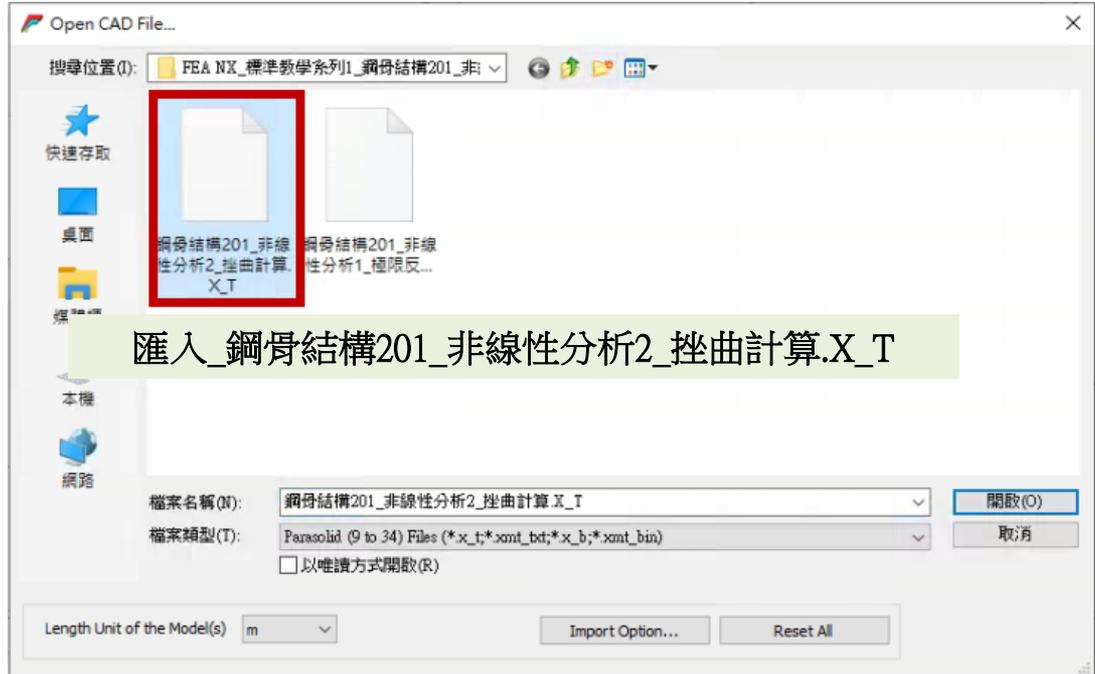
DWG (Wireframe)...
Import the DWG file into a current project

midas Mxt...
Import the midas MXT file.

GeoXD Neutral Format File(*.FPN)...
Import the Neutral File

GTS NX Neutral Format...
Open Neutral File

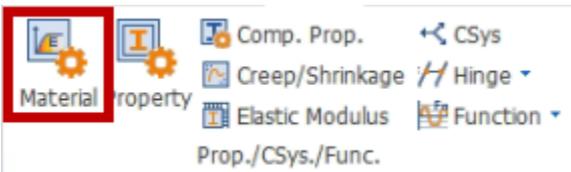
Import Nodal Results(*.txt)
Import nodal results File



鋼構立柱3D模型



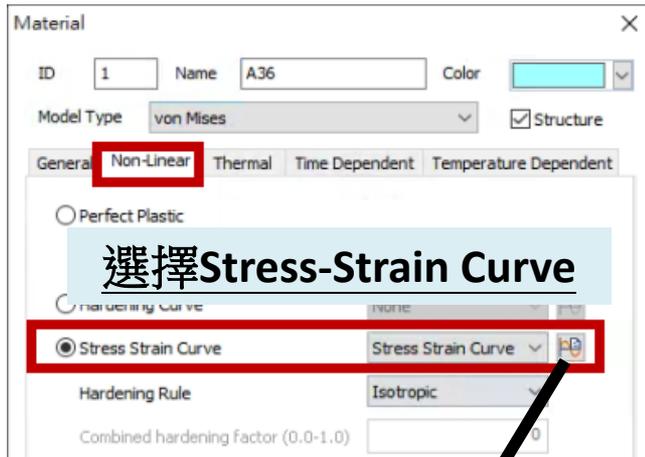
材料



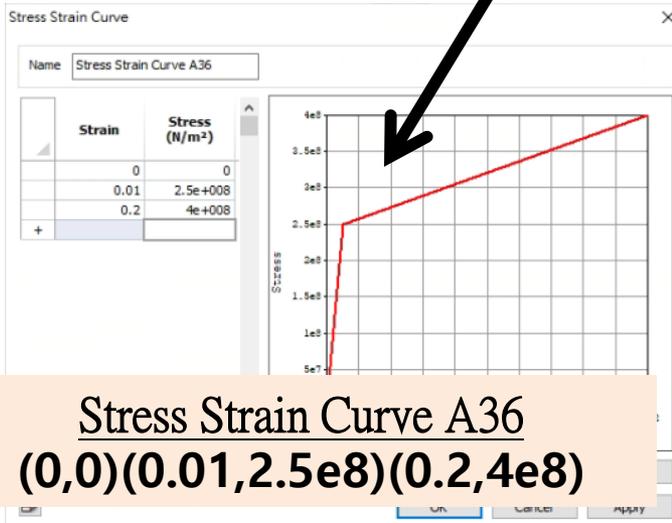
自訂材料名稱A36

材料模型: Von Mises

勾選Structure



選擇Stress-Strain Curve

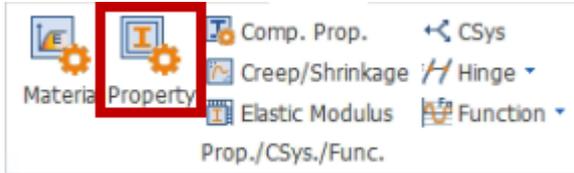


Stress Strain Curve A36
(0,0)(0.01,2.5e8)(0.2,4e8)

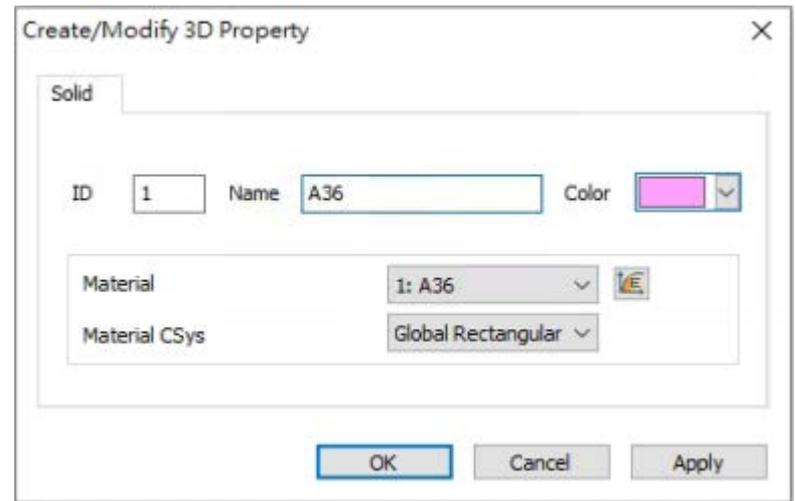
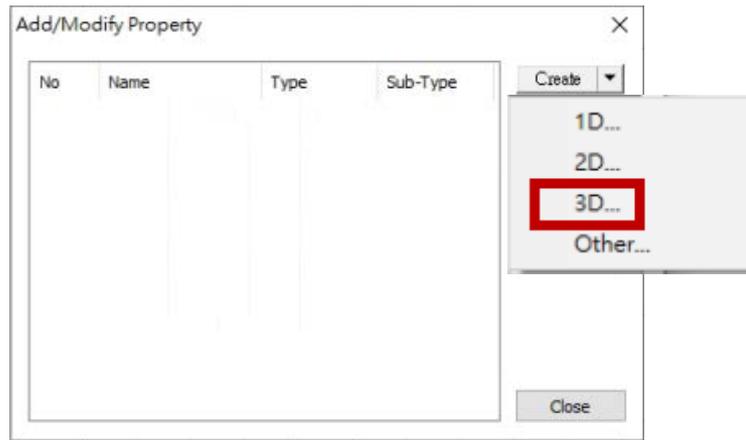
Steel/ASTM(S)/A36

註:未勾選Structure則視為土壤材料。

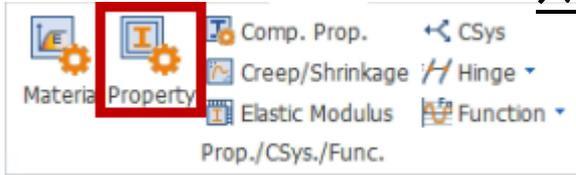
屬性



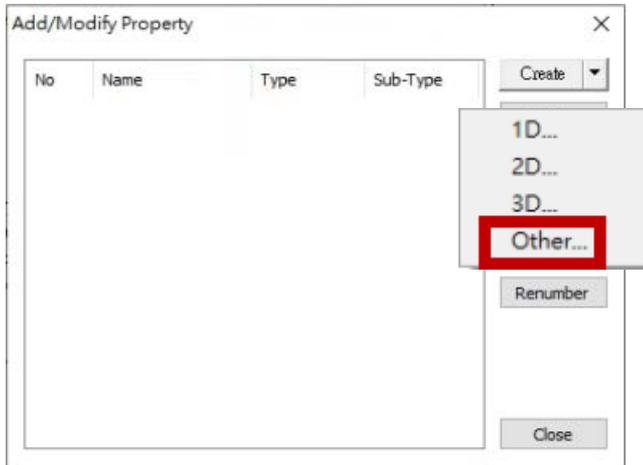
新增A36 Property



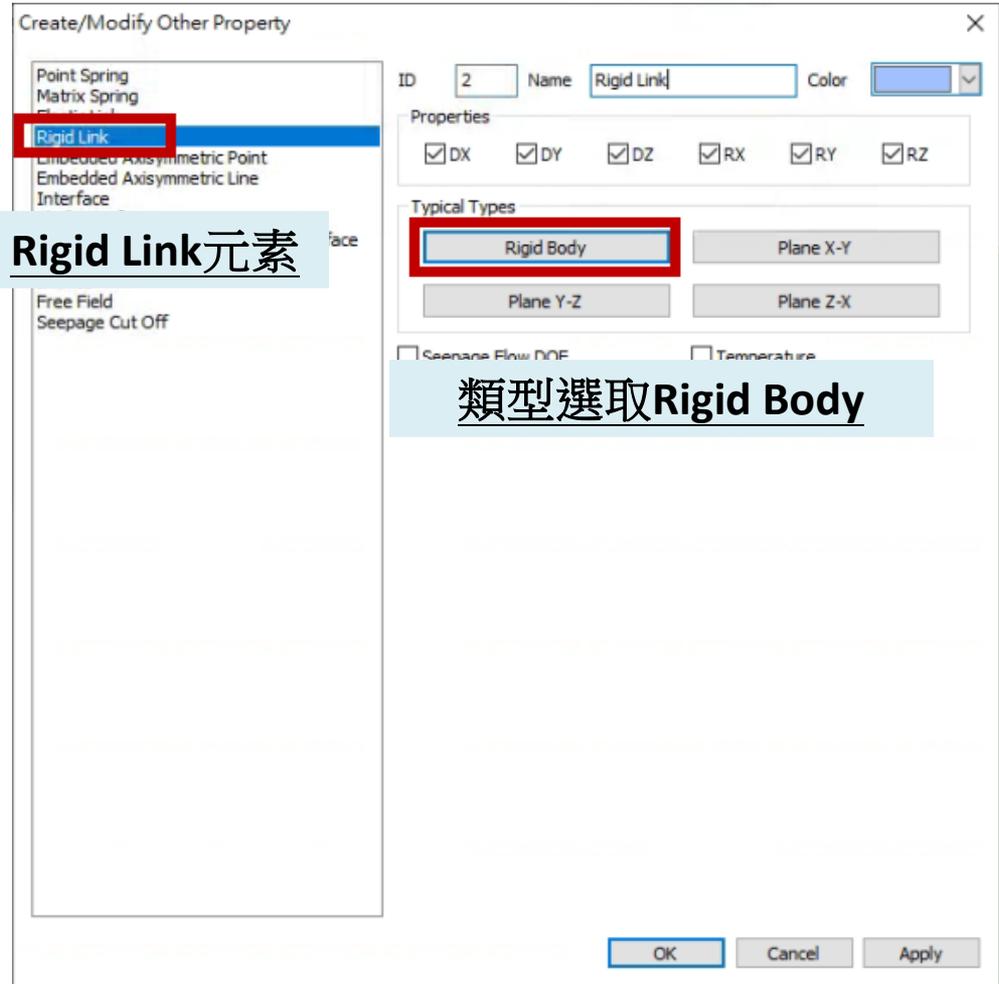
屬性-Rigid Link



新增Rigid Link Property



Rigid Link元素

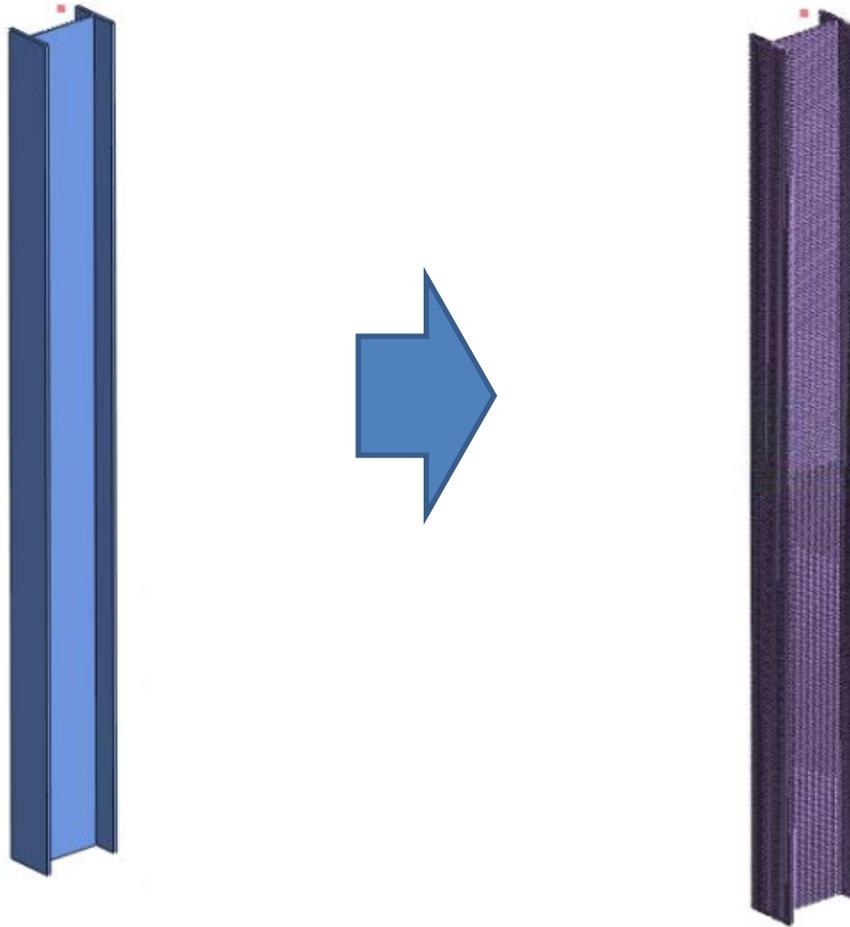


類型選取Rigid Body

3D網格-混合網格

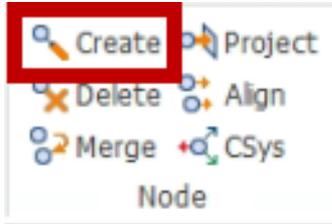


Hybrid Mesher(混合網格)/網格尺寸0.015 (m)

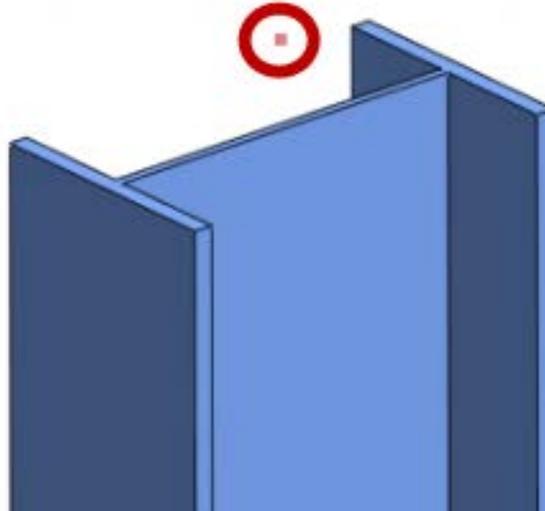


主控節點建立

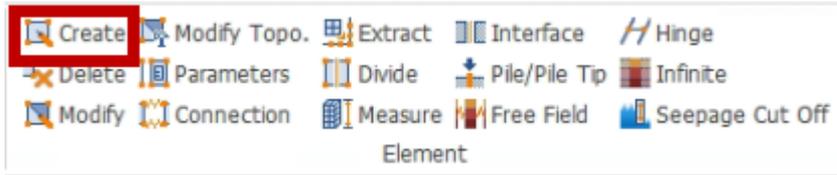
(剛性連接主控點)



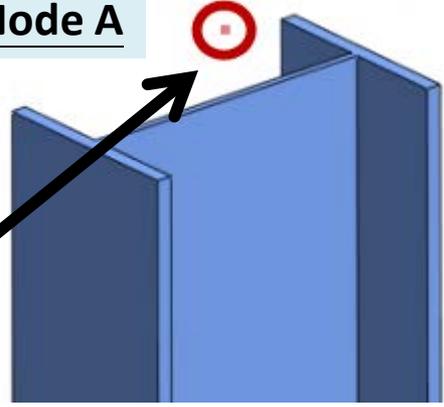
Node A



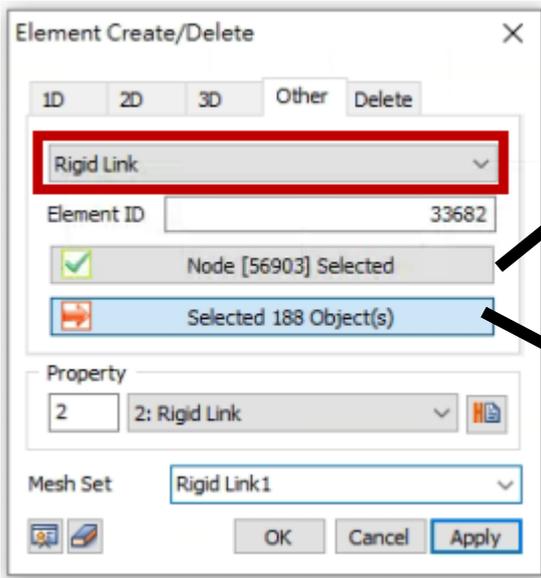
剛性連接



Node A



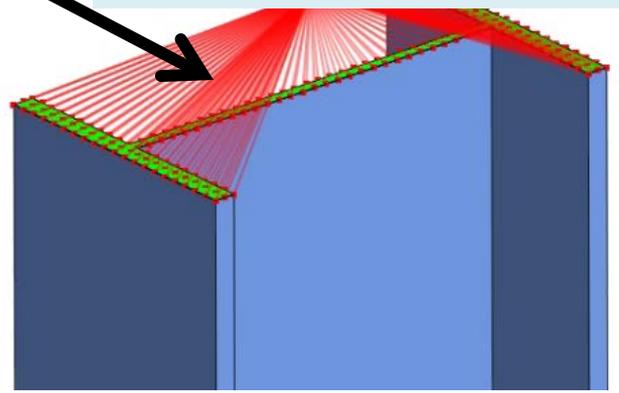
元素類型選取Rigid Link



切換使用面特徵選取節點

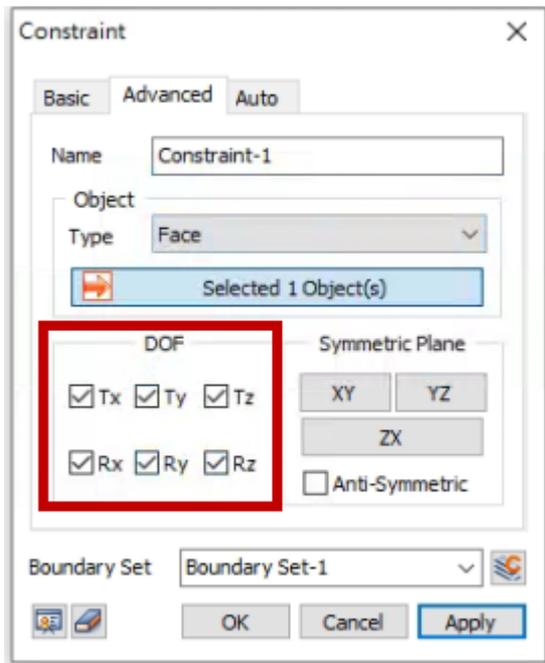


Node A側周邊面特徵節點



自訂網格集Rigid Link1

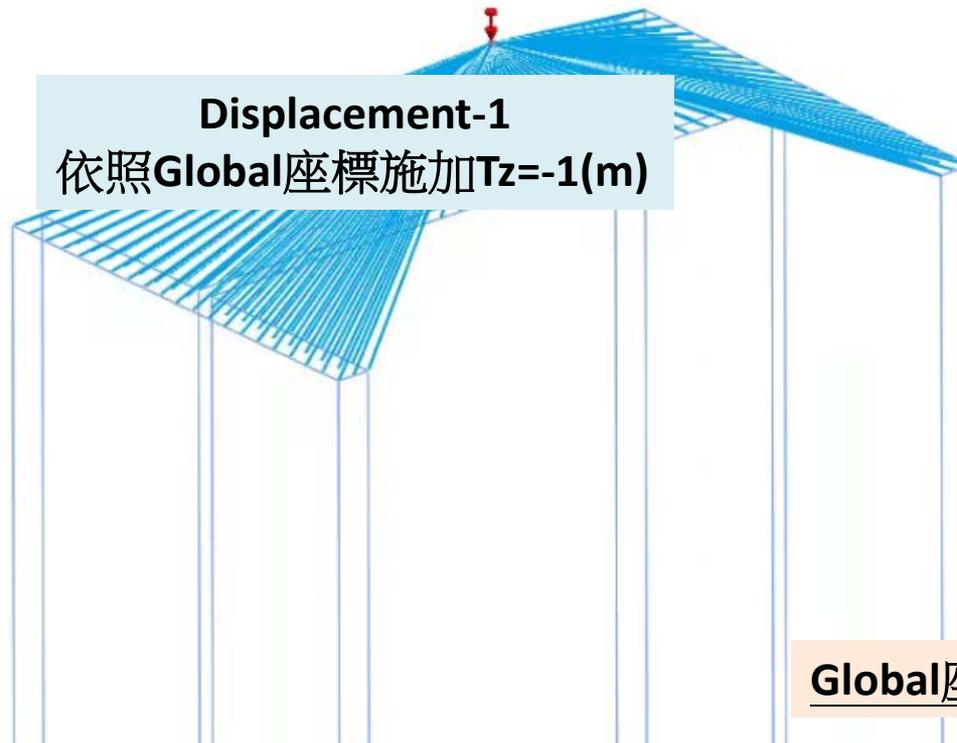
邊界



立柱底部面特徵
Tx/Ty/Tz/Rx/Ry/Rz



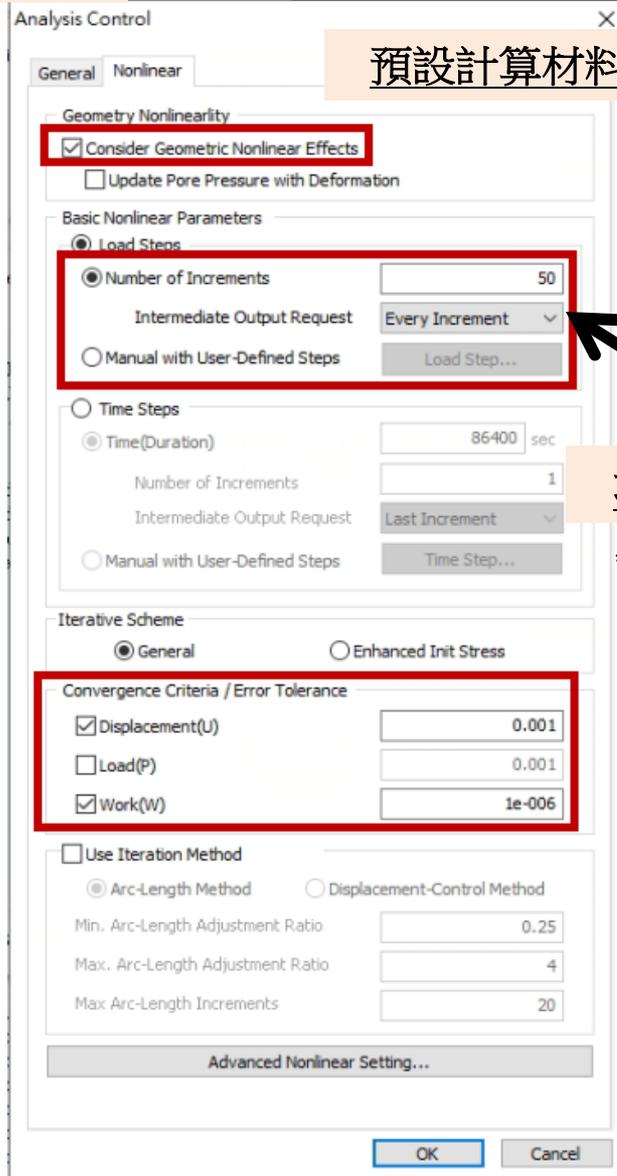
位移



非線性分析-1



非線性分析-2



預設計算材料非線性,勾選才計算幾何&邊界(接觸)非線性

增量數分成50等份/輸出每次增量結果
每一增量增加變形量 $1(m)/50=0.02(m)$

收斂判斷採Displacement/Work

註:後挫曲計算需考慮幾何非線性。

非線性分析-3

輸出控制

Output Control

Output Type Output Option

Write Results of All Active Mesh Sets

Nodal Results

Displacement Mesh Set...

Applied Load Mesh Set...

Reaction Force Mesh Set...

Grid Point Force Mesh Set...

Contact Mesh Set...

Element Results

Force Mesh Set...

Stress Mesh Set...

Strain Mesh Set...

Status Mesh Set...

Ductility Mesh Set...

Multi-layered Grid Mesh Set...

Element Output Location

Element Corner Results

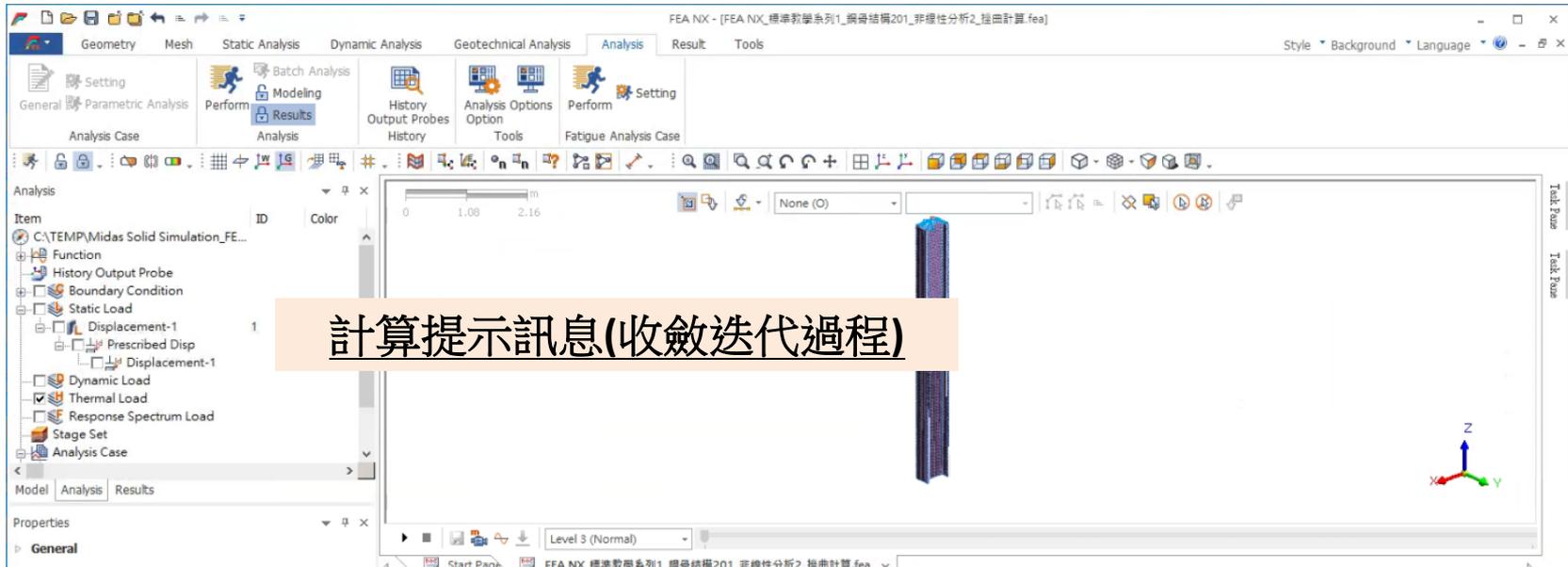
Shell Mid-Plane Results

Number of Beam Output Segments 4

OK Cancel

內力/反力結果輸出

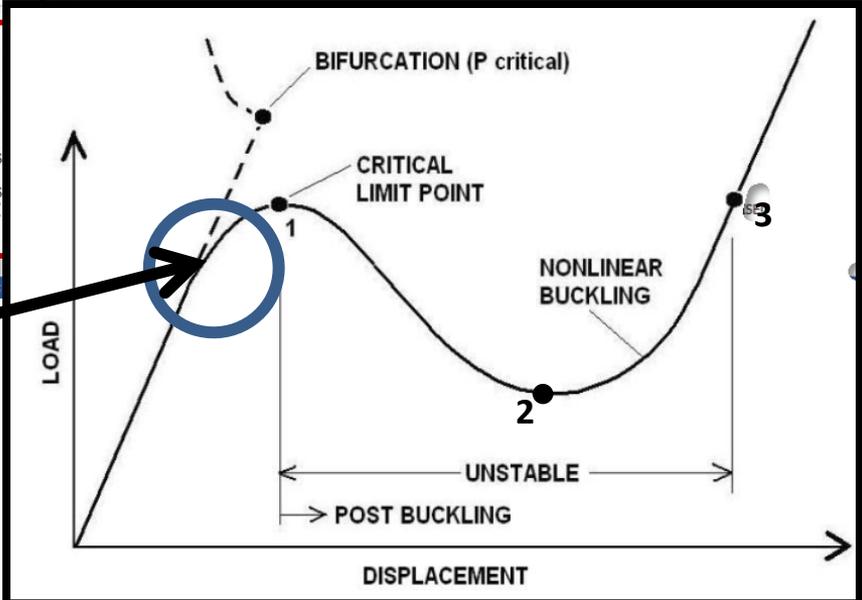
執行分析



計算提示訊息(收斂迭代過程)

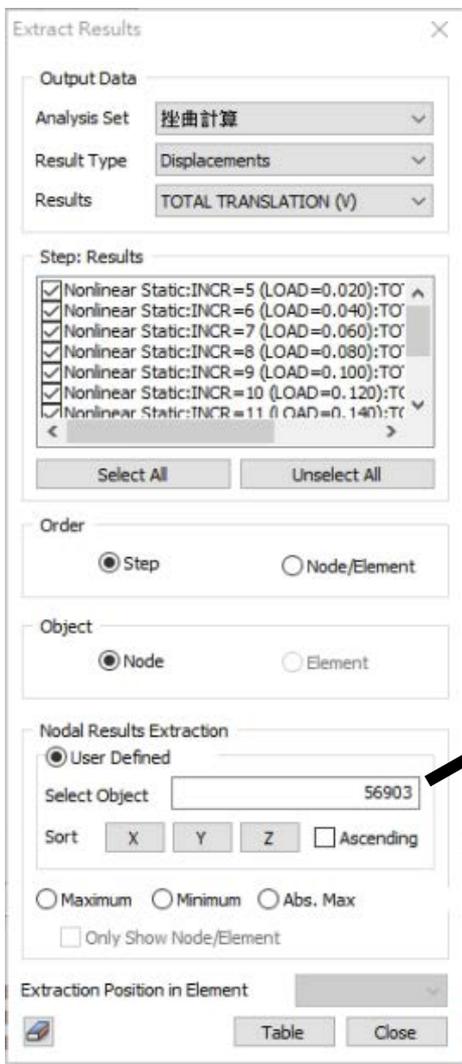
```
Output
> INCREMENT= 17 ( 20.50%), ITERATION= 18, ERROR NORMS: U(
> INCREMENT= 18 ( 21.00%), ITERATION= 3, ERROR NORMS: U(
...
> INCREMENT= 18 ( 20.75%), ITERATION= 14, ERROR NORMS: U(
> INCREMENT= 18 ( 20.62%), ITERATION= 2, ERROR NORMS: U(
> INCREMENT= 19 ( 20.56%), ITERATION= 28, ERROR NORMS: U(
> WARNING [4024] : FAILED TO CONVERGE IN NONLINEAR ANALYSIS
> INCREMENT= 19 ( 20.62%), ITERATION= 25, ERROR NORMS: U(
> WARNING [4024] : FAILED TO CONVERGE IN NONLINEAR ANALYSIS
> INCREMENT= 20 ( 20.75%), ITERATION= 0, ERROR NORMS: U(
> INCREMENT= 20 ( 20.69%), ITERATION= 1, ERROR NORMS: U(
> ERROR [2103] : FAILED TO CONVERGE IN NONLINEAR ANALYSIS.
```

計算不收斂(終止計算)
對照挫曲位移-載荷曲線

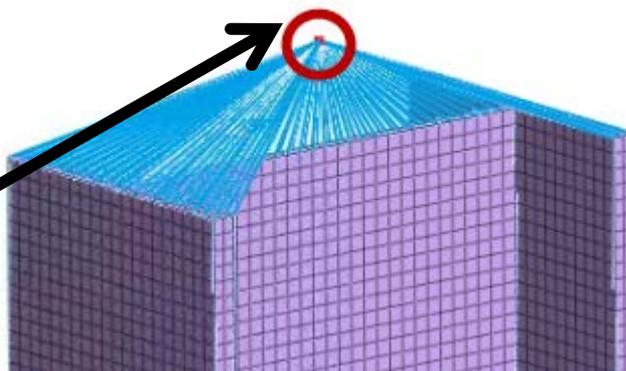


分析結果-1

迭代過程剛性中心點位移結果



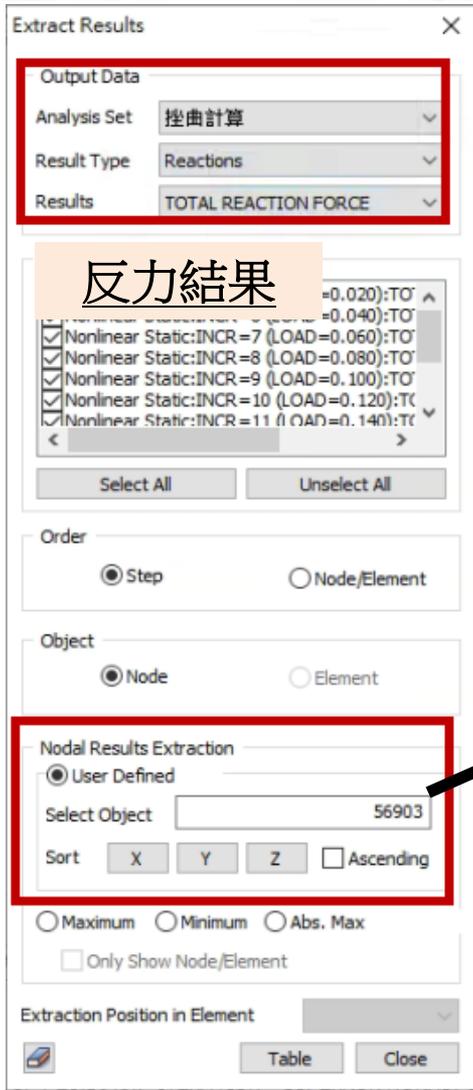
No	Step	Step Value	Node:56903 TOTAL TRANSLATION (V) (m)
1	Nonlinear Static:INCR=5 (LOAD=0.020)	2.000000e-002	2.000000e-002
2	Nonlinear Static:INCR=6 (LOAD=0.040)	4.000000e-002	4.000000e-002
3	Nonlinear Static:INCR=7 (LOAD=0.060)	6.000000e-002	6.000000e-002
4	Nonlinear Static:INCR=8 (LOAD=0.080)	8.000000e-002	8.000000e-002
5	Nonlinear Static:INCR=9 (LOAD=0.100)	1.000000e-001	1.000000e-001
6	Nonlinear Static:INCR=10 (LOAD=0.120)	1.200000e-001	1.200000e-001
7	Nonlinear Static:INCR=11 (LOAD=0.140)	1.400000e-001	1.400000e-001
8	Nonlinear Static:INCR=12 (LOAD=0.160)	1.600000e-001	1.600000e-001
9	Nonlinear Static:INCR=13 (LOAD=0.180)	1.800000e-001	1.800000e-001
10	Nonlinear Static:INCR=14 (LOAD=0.200)	2.000000e-001	2.000000e-001



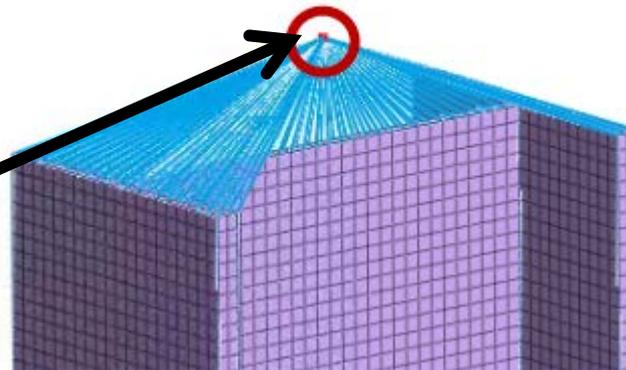
提取-迭代過程結果

分析結果-2

迭代過程剛性中心點反力結果

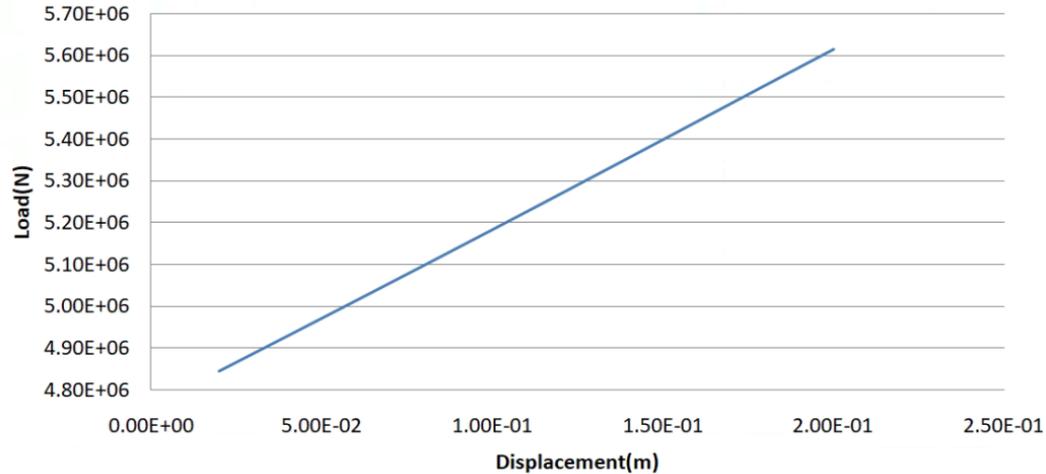


No	Step	Step Value	Node:56903 TOTAL REACTION FORCE (N)
1	Nonlinear Static:INCR=5 (LOAD=0.020)	2.000000e-002	4.844010e+006
2	Nonlinear Static:INCR=6 (LOAD=0.040)	4.000000e-002	4.928154e+006
3	Nonlinear Static:INCR=7 (LOAD=0.060)	6.000000e-002	5.012475e+006
4	Nonlinear Static:INCR=8 (LOAD=0.080)	8.000000e-002	5.097251e+006
5	Nonlinear Static:INCR=9 (LOAD=0.100)	1.000000e-001	5.182576e+006
6	Nonlinear Static:INCR=10 (LOAD=0.120)	1.200000e-001	5.268476e+006
7	Nonlinear Static:INCR=11 (LOAD=0.140)	1.400000e-001	5.354899e+006
8	Nonlinear Static:INCR=12 (LOAD=0.160)	1.600000e-001	5.441698e+006
9	Nonlinear Static:INCR=13 (LOAD=0.180)	1.800000e-001	5.528546e+006
10	Nonlinear Static:INCR=14 (LOAD=0.200)	2.000000e-001	5.614708e+006



分析結果-3

Buckling (Displacement-Load Curve)



No	Step	Step Value
1	Nonlinear Static:INCR=5 (LOAD=0.020)	2.000000e-002
2	Nonlinear Static:INCR=6 (LOAD=0.040)	4.000000e-002
3	Nonlinear Static:INCR=7 (LOAD=0.060)	6.000000e-002
4	Nonlinear Static:INCR=8 (LOAD=0.080)	8.000000e-002
5	Nonlinear Static:INCR=9 (LOAD=0.100)	1.000000e-001
6	Nonlinear Static:INCR=10 (LOAD=0.120)	1.200000e-001
7	Nonlinear Static:INCR=11 (LOAD=0.140)	1.400000e-001
8	Nonlinear Static:INCR=12 (LOAD=0.160)	1.600000e-001
9	Nonlinear Static:INCR=13 (LOAD=0.180)	1.800000e-001
10	Nonlinear Static:INCR=14 (LOAD=0.200)	2.000000e-001

Node:56903 TOTAL TRANSLATION (V) (m)	
	2.000000e-002
	4.000000e-002
	6.000000e-002
	8.000000e-002
	1.000000e-001
	1.200000e-001
	1.400000e-001
	1.600000e-001
	1.800003e-001
	2.000009e-001

X軸

No	Step	Step Value
1	Nonlinear Static:INCR=5 (LOAD=0.020)	2.000000e-002
2	Nonlinear Static:INCR=6 (LOAD=0.040)	4.000000e-002
3	Nonlinear Static:INCR=7 (LOAD=0.060)	6.000000e-002
4	Nonlinear Static:INCR=8 (LOAD=0.080)	8.000000e-002
5	Nonlinear Static:INCR=9 (LOAD=0.100)	1.000000e-001
6	Nonlinear Static:INCR=10 (LOAD=0.120)	1.200000e-001
7	Nonlinear Static:INCR=11 (LOAD=0.140)	1.400000e-001
8	Nonlinear Static:INCR=12 (LOAD=0.160)	1.600000e-001
9	Nonlinear Static:INCR=13 (LOAD=0.180)	1.800000e-001
10	Nonlinear Static:INCR=14 (LOAD=0.200)	2.000000e-001

Node:56903 TOTAL REACTION FORCE (N)	
	4.844010e+006
	4.928154e+006
	5.012475e+006
	5.097751e+006
	5.183027e+006
	5.268303e+006
	5.353579e+006
	5.438855e+006
	5.524131e+006
	5.609407e+006

Y軸

FEA NX_標準教學系列

鋼骨結構-非線性分析
後挫曲分析

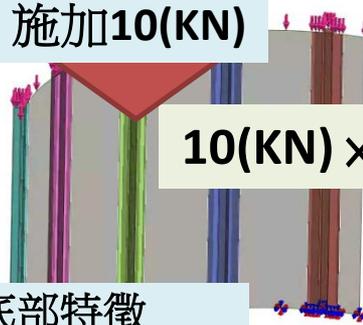
台灣邁達斯

線性挫曲 & 非線性挫曲

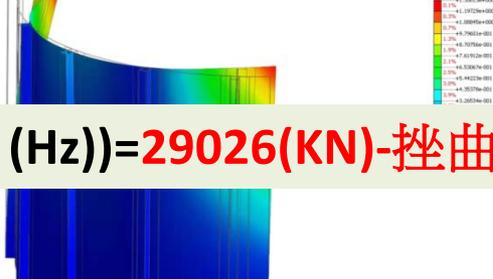
(Linear & Non-Linear Buckling)

FEA NX-線性挫曲分析(挫曲最大臨界力)

施加10(KN)



$10(\text{KN}) \times 2902.6(\text{mode 1 (Hz)}) = 29026(\text{KN})$ - 挫曲最大臨界力



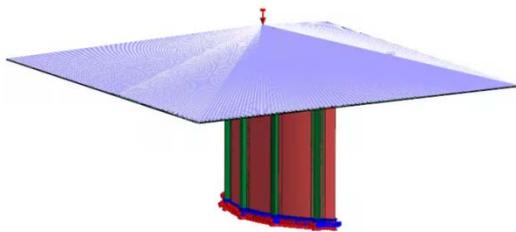
Mode 1-變形

底部特徵

Tx/Ty/Tz/Rx/Ry/Rz

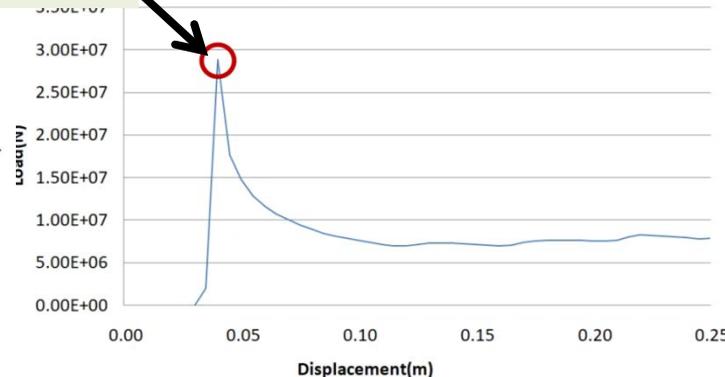
FEA NX-非線性挫曲分析(挫曲過程載荷-位移曲線)

下壓0.25(m)



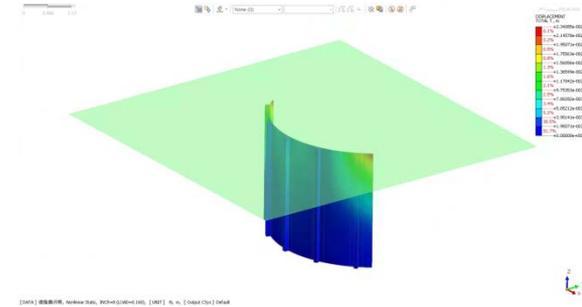
28832.9(KN)

Displacement Load Curve



底部特徵

Tx/Ty/Tz/Rx/Ry/Rz



後挫曲

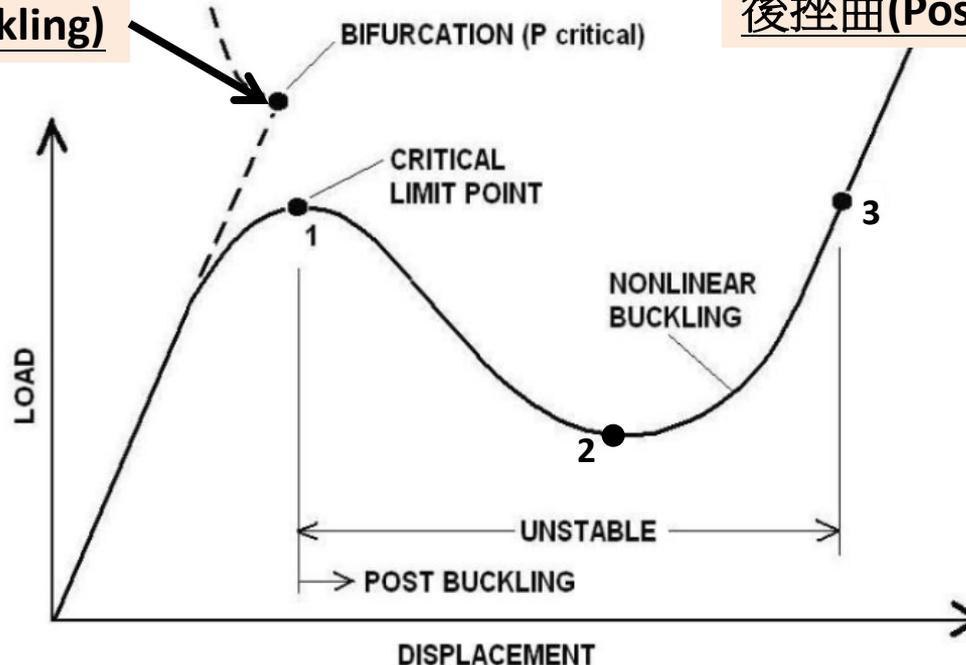
後挫曲(非線性挫曲)分析

載荷較小時分析結果與線性結果一致。當載荷逐步增大，變形逐步增加，結構響應呈現出非線性，力位移曲線開始偏離線性結果，在Point 1點承受外部載荷達到最大。

之後變形繼續變大但是結構能承受的外部載荷變小，結構發生挫曲，變形繼續變大，但是結構承受的載荷繼續變小。達到Point 2位移增加，結構承受的載荷開始增加，Point 3達到之前載荷的極限點。從Point 1到Point 3的過程稱為疾速跳過(snap through)。

線性挫曲(Linear Buckling)

後挫曲(Post Buckling)



Reference

<https://www.linkedin.com/pulse/buckling-post-buckling-abraham-imam-muttaqin>

應變硬化指數

(Strain Hardening Exponent)

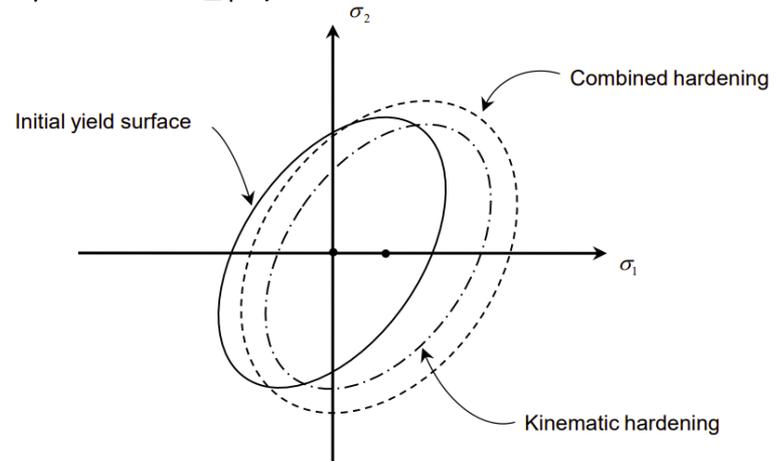
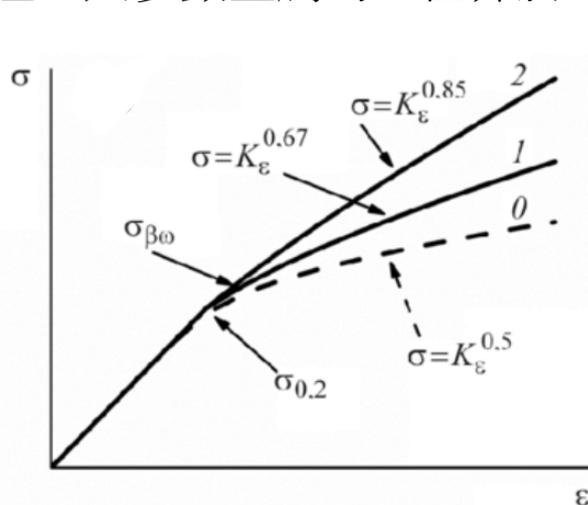
應變硬化指數(Strain Hardening Exponent)

通常表示為 n ，常用於與加工硬化中的應力-應變行為相關的計算中的常量。

$$\sigma = K \epsilon^n$$

- σ 真實應力
- ϵ 真實應變
- K 強度係數
- n 應變硬化指數

應變硬化指數(n), 值介於 0 和 1 之間，0 表示完全塑性，1 表示完全彈性，大多數金屬的 n 值介於 0.10 和 0.50 之間。



硬化模型-屈服面

Reference

GTS NX AND FEA NX Manual

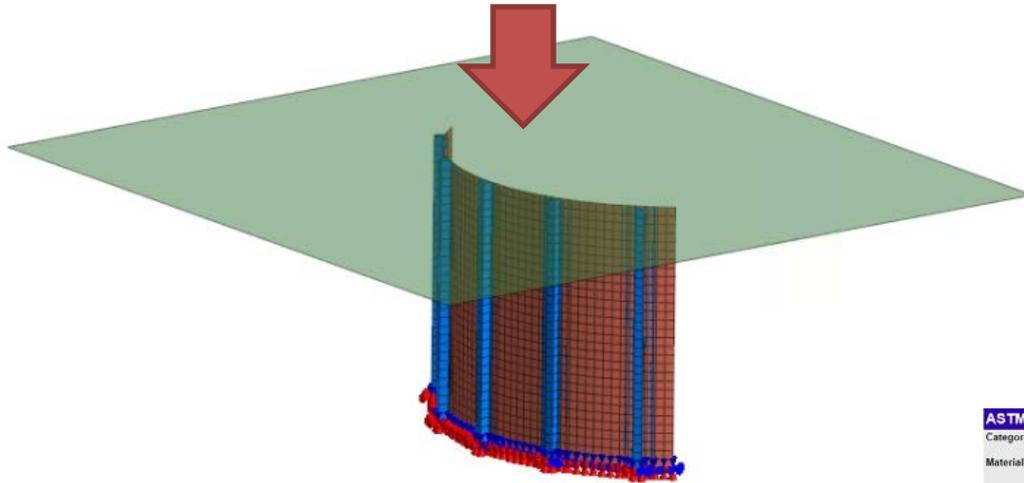
https://en.wikipedia.org/wiki/Strain_hardening_exponent

分析說明

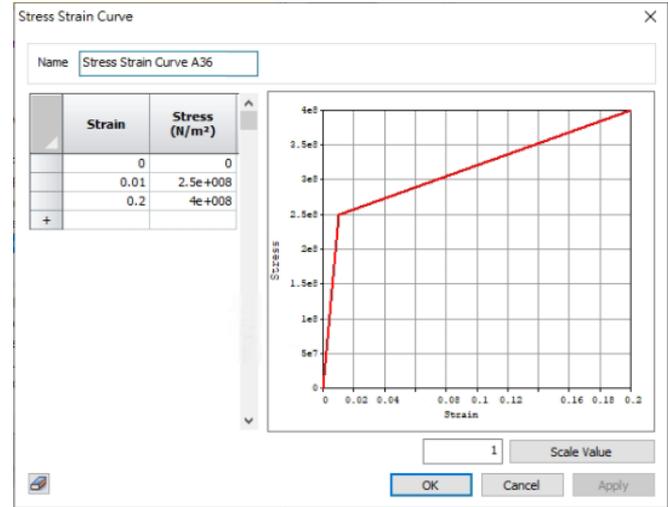
計算下壓過程位移過程所需載荷!!

材料Steel/ASTM A36

鋼板下壓0.2(m)



底部節點
Tx/Ty/Tz/Rx/Ry/Rz



ASTM A36 Steel, plate

Categories: Metal, Ferrous Metal, ASTM Steel, Carbon Steel, Low Carbon Steel

Material Notes: Steel for general structural purposes including bridges and buildings. Minimum Cu content when copper steel is specified. Tests performed in transverse direction for plates wider than 590 mm.

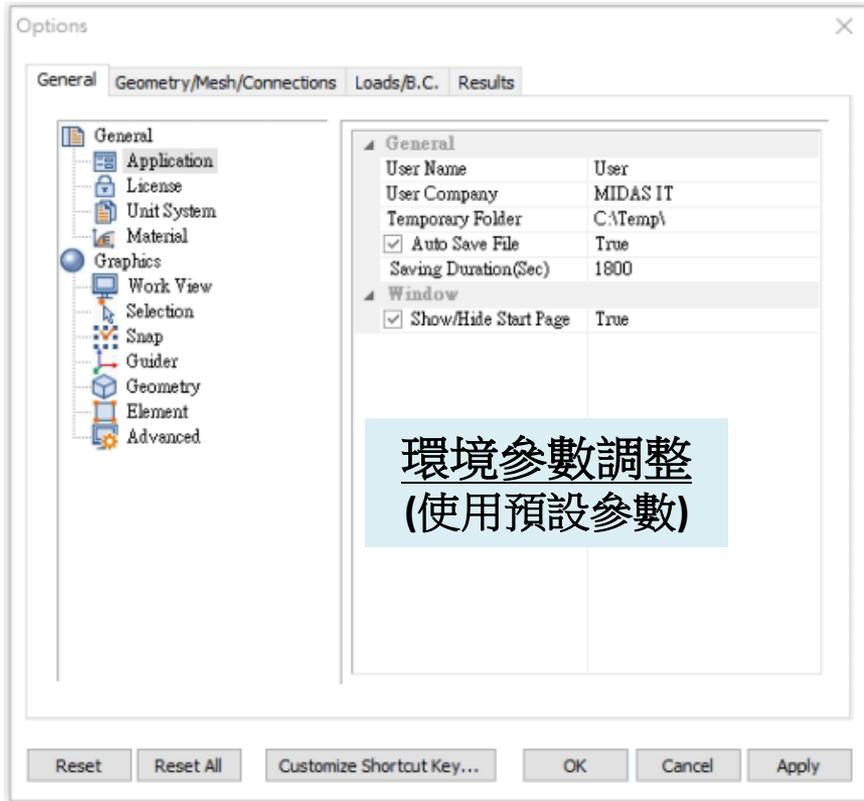
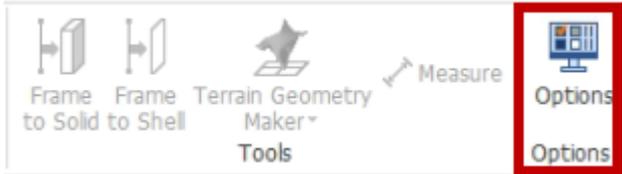
Key Words: UNS K02600

Vendors: No vendors are listed for this material. Please [click here](#) if you are a supplier and would like information on how to add your listing to this material.

[Print friendly version](#) [Download as PDF](#) [Download to Excel \(requires Excel and Windows\)](#)
[Export data to your CAD/FEA program](#)

Physical Properties	Metric	English
Density	7.80 g/cc	0.282 lb/in³
Mechanical Properties	Metric	English
Tensile Strength, Ultimate	400 - 550 MPa	58000 - 79000 psi
Tensile Strength, Yield	250 MPa	36300 psi
Elongation at Break	20 %	20 %
	23 %	23 %
Modulus of Elasticity	200 GPa	29000 ksi
Bulk Modulus	160 GPa	23200 ksi
Poissons Ratio	0.26	0.26
Shear Modulus	79.3 GPa	11500 ksi
Component Elements Properties	Metric	English
Carbon, C	0.25 - 0.29 %	0.25 - 0.29 %
Copper, Cu	0.20 %	0.20 %
Iron, Fe	98 %	98 %
Manganese, Mn	1.03 %	1.03 %
Phosphorus, P	<= 0.040 %	<= 0.040 %
Silicon, Si	0.28 %	0.28 %
Sulfur, S	<= 0.050 %	<= 0.050 %

環境

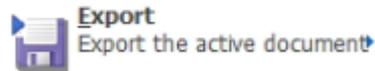


新文件



單位使用N/m/J/sec

匯入模型



Import the geometries or FE model in the selected file

- Import CAD File...**
Import the CAD file into a current project
- DXF 2D (Wireframe)...**
Import the DXF 2D file into a current project
- DXF 3D (Wireframe)...**
Import the DXF 3D file into a current project
- DWG (Wireframe)...**
Import the DWG file into a current project
- midas Mxt...**
Import the midas MXT file.
- GeoXD Neutral Format File(*.FPN)...**
Import the Neutral File
- GTS NX Neutral Format...**
Open Neutral File
- Import Nodal Results(*.txt)**
Import nodal results File

Open CAD File...

搜尋位置 (I): FEA NX_標準教學系列1_鋼骨結構201_非...

檔案名稱 (N): 鋼骨結構201_非線性分析3_後挫曲分析.X_T

檔案類型 (T): Parasolid (9 to 34) Files (*.x_t;*.xmt_btx;*.x_b;*.xmt_bin)

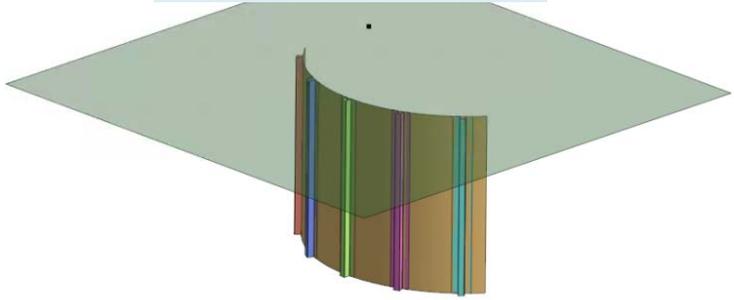
以唯讀方式開啟 (R)

Length Unit of the Model(s) m

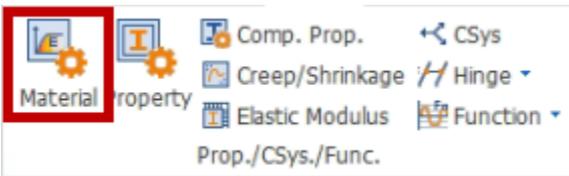
Import Option... Reset All

Buttons: 開啟 (O), 取消

鋼結構3D模型



材料-1



Material ID 1
Model Type **von Mises** Structure

材料模型: Von Mises

勾選 Structure

Material ID 1 Name A36 Color
Model Type von Mises Structure

選擇 Stress-Strain Curve
(硬化指數: 0.45)

Stress Strain Curve
Hardening Rule Isotropic+Kinematic
Combined hardening factor (0.0-1.0) 0.45

Inc. of Elastic Modulus Ref. Height 0 m
Poisson's Ratio(v) 0.3
Unit Weight(γ) 77091.0111 N/m³

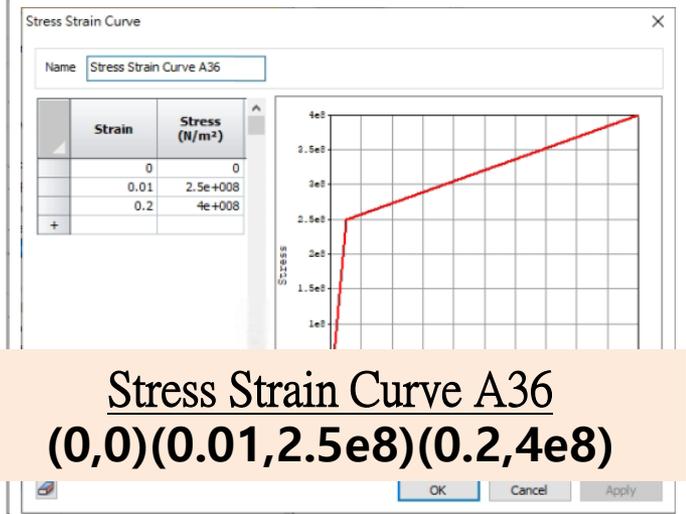
Initial Stress Parameters
Ko Determination 1
 Automatic
 Manual Anisotropy

Thermal Parameter
Thermal Coefficient 1.17e-005 1/[T]
Molecular vapor diffusion coefficient 0 m²/sec
Thermal diffusion enhancement 0

Damping Ratio(For Dynamic)
Damping Ratio 0.05

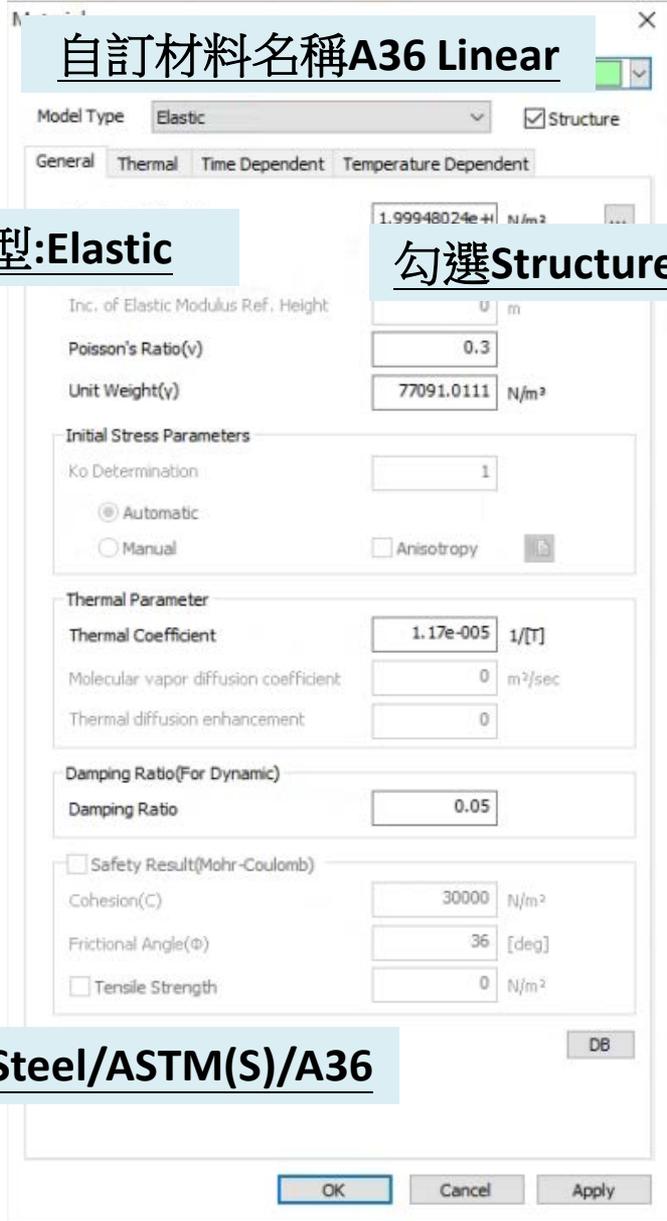
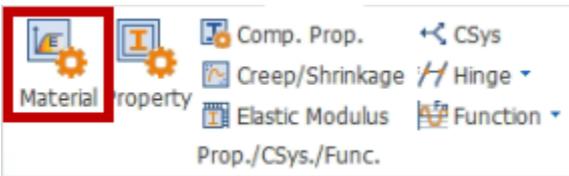
Safety Result(Mohr-Coulomb)
Cohesion(C) 30000 N/m²
Frictional Angle(ϕ) 36 [deg]
 Tensile Strength 0 N/m²

Steel/ASTM(S)/A36 DB



Stress Strain Curve A36
(0,0)(0.01,2.5e8)(0.2,4e8)

材料-2

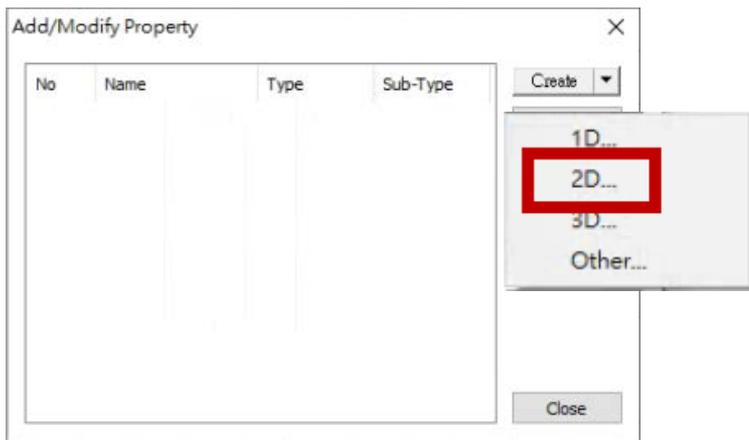
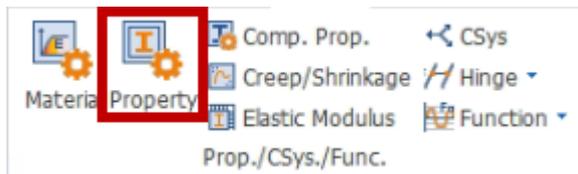


材料模型:Elastic

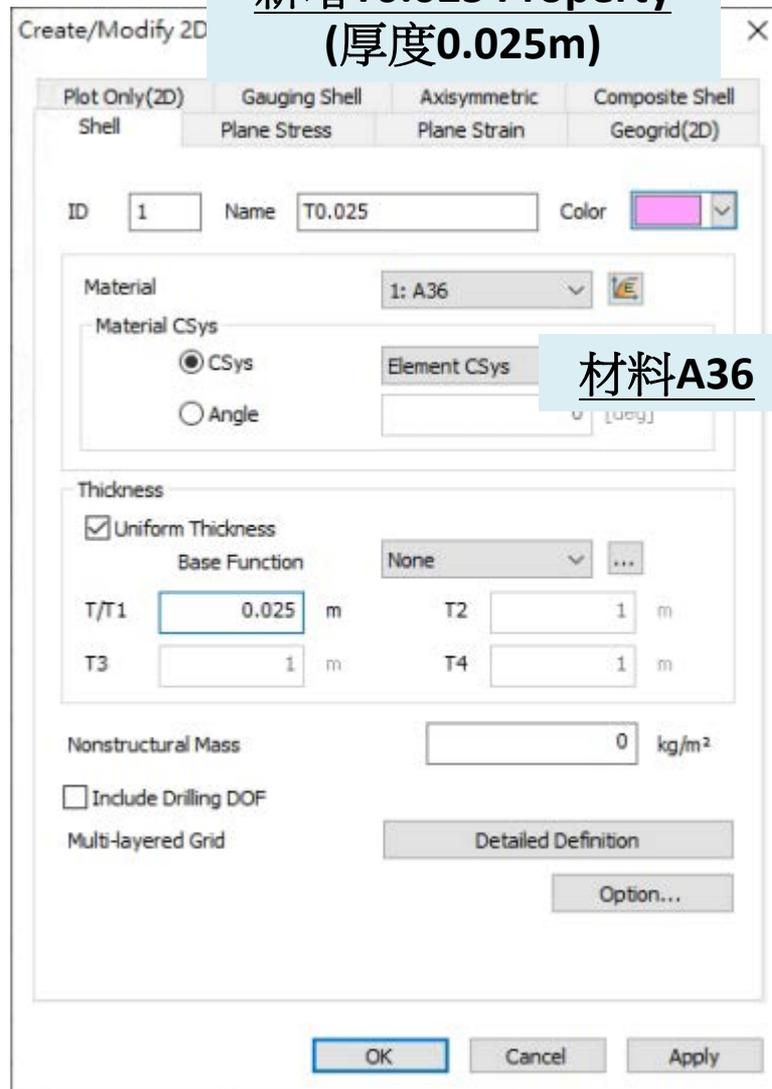
勾選Structure

Steel/ASTM(S)/A36

屬性-2D-T0.025

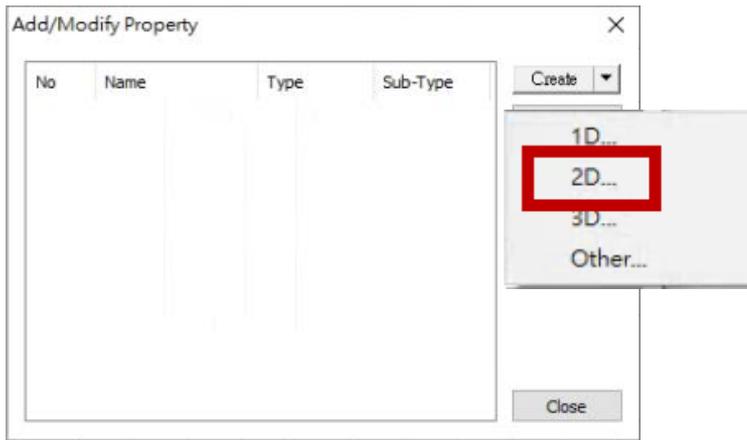
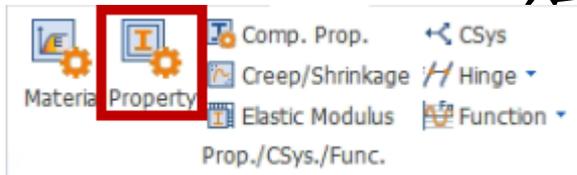


新增T0.025 Property
(厚度0.025m)

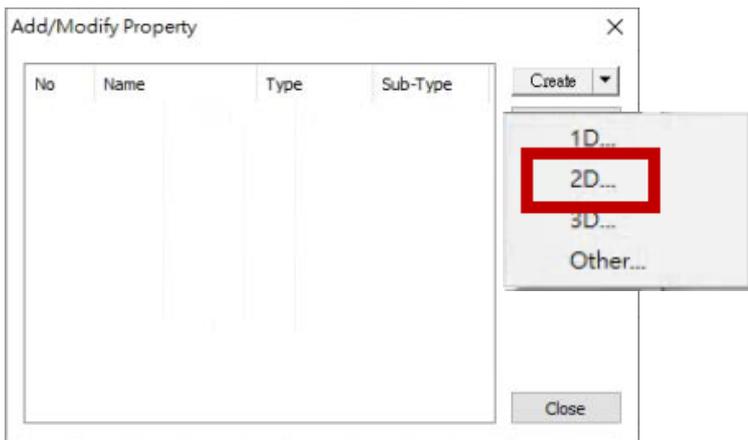
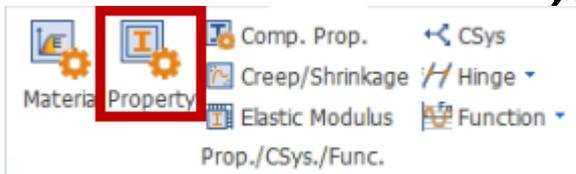


材料A36

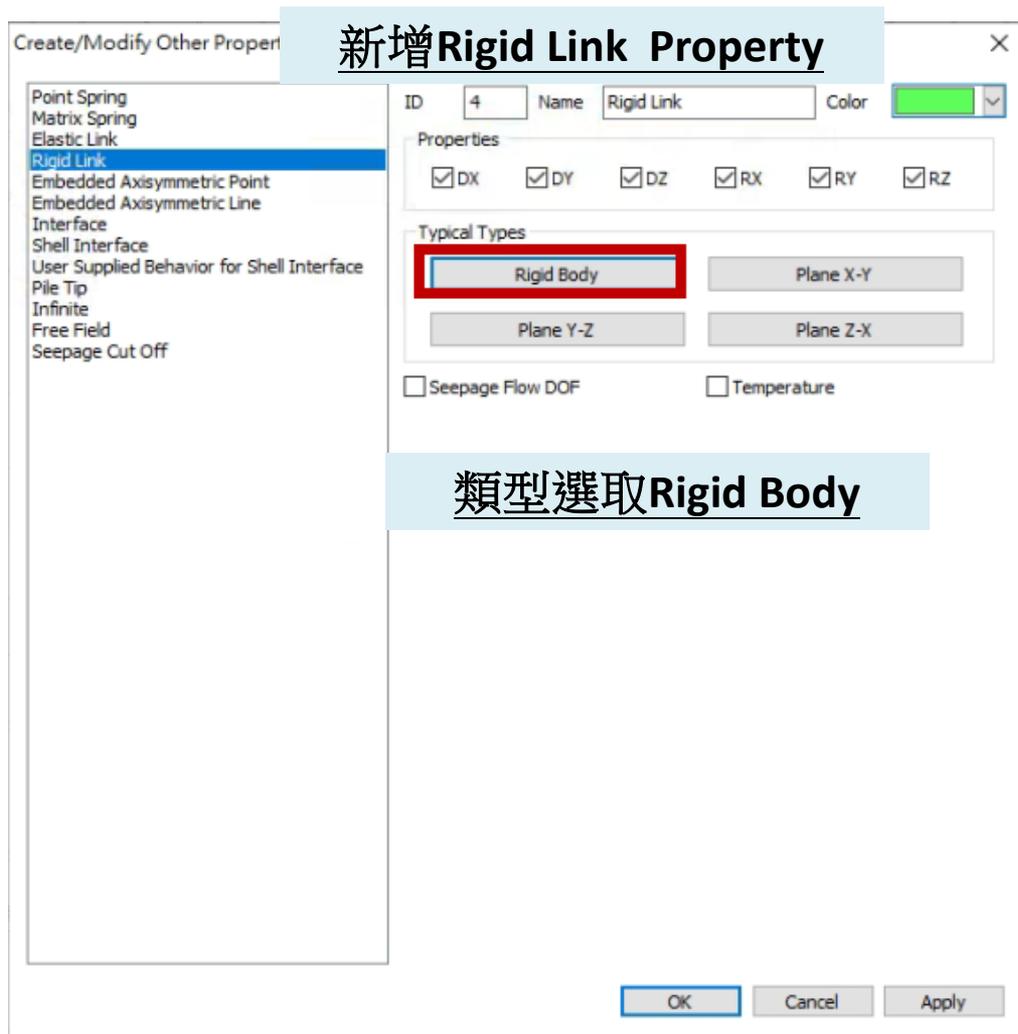
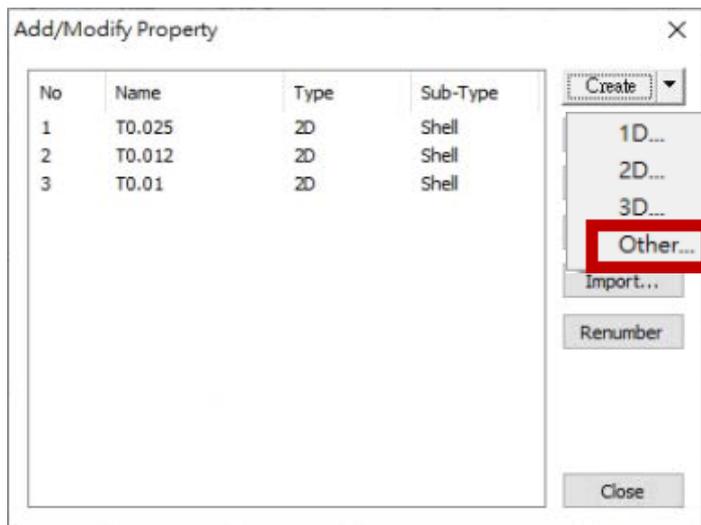
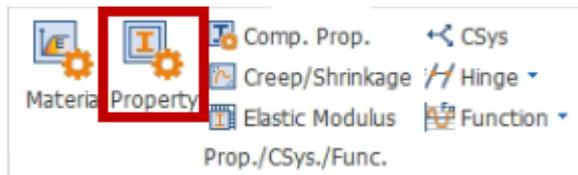
屬性-2D-T0.012



屬性-2D-T0.01



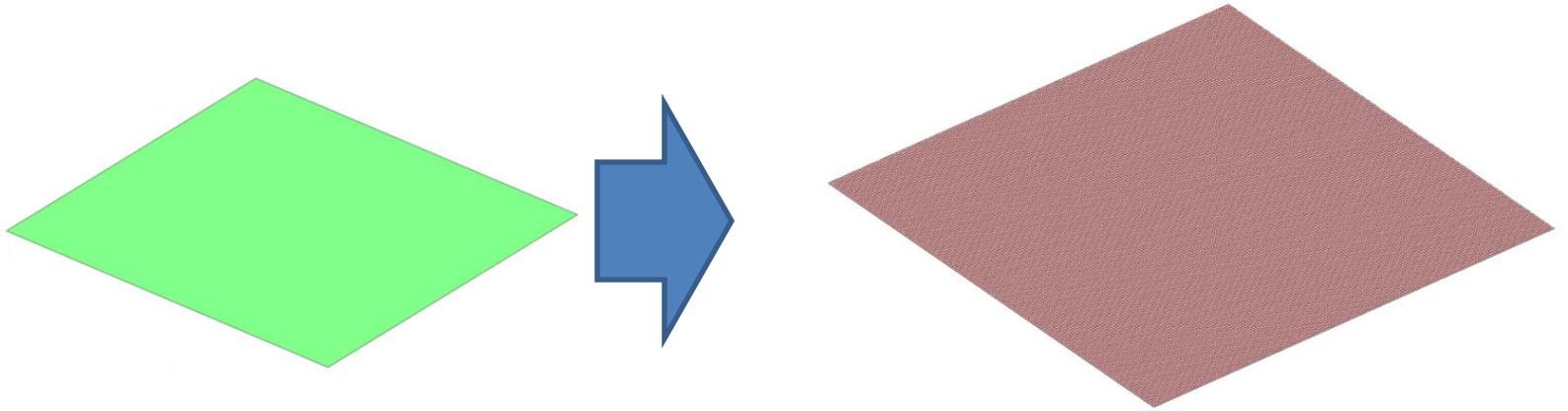
屬性-Rigid Link



2D網格-下壓板

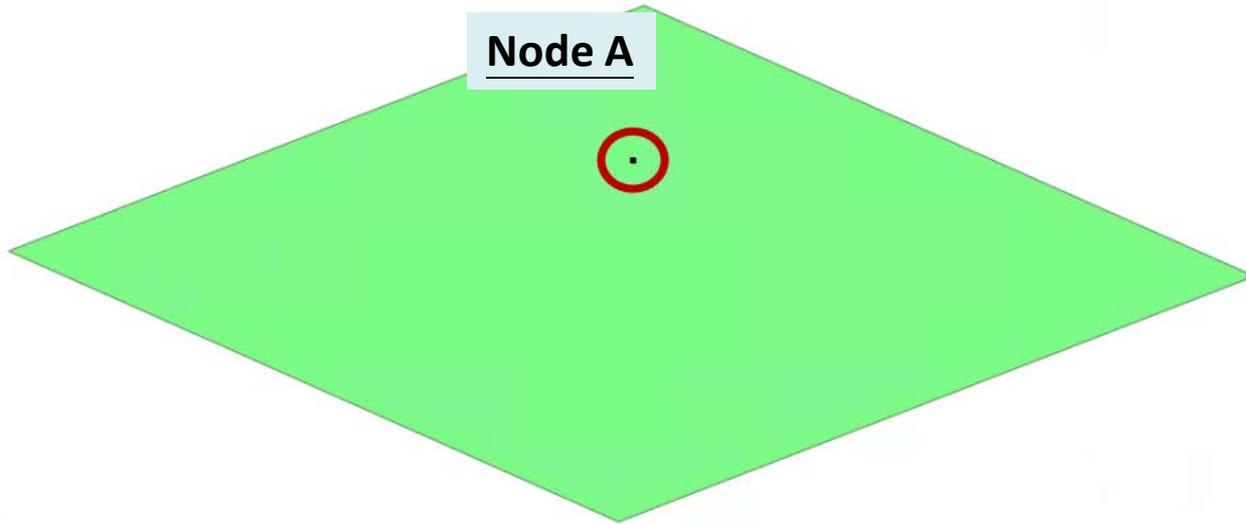
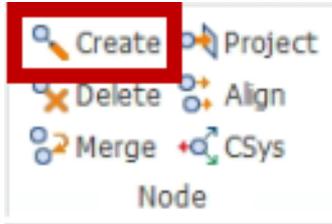


網格尺寸0.1 (m)/屬性T0.01/自訂網格集名稱(下壓板)

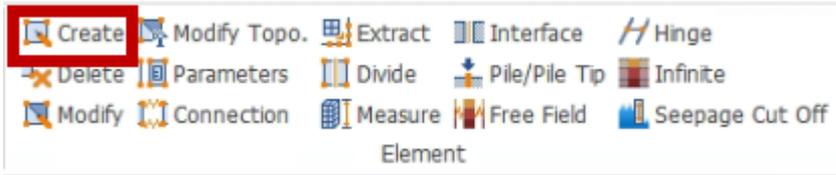


主控節點建立

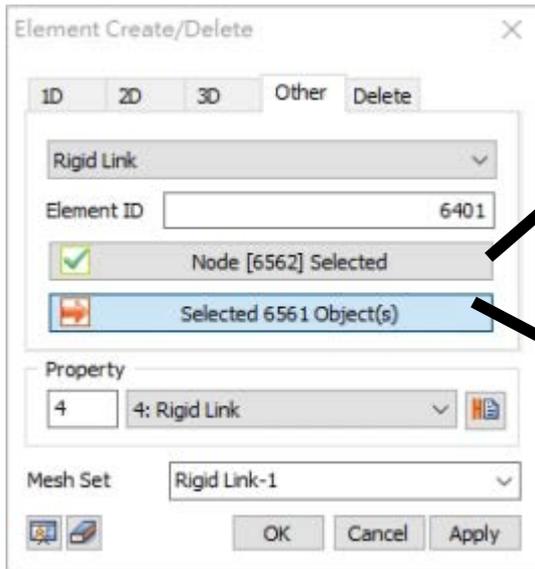
(剛性連接主控點)



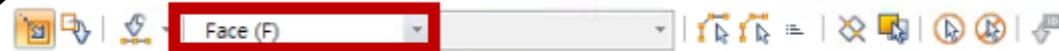
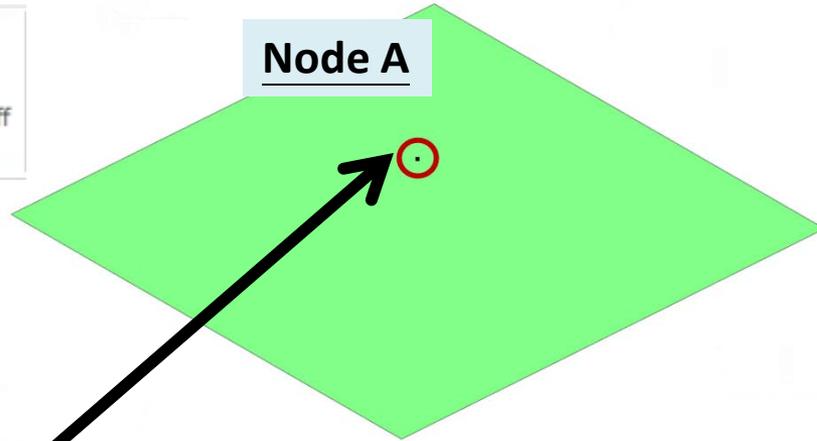
剛性連接



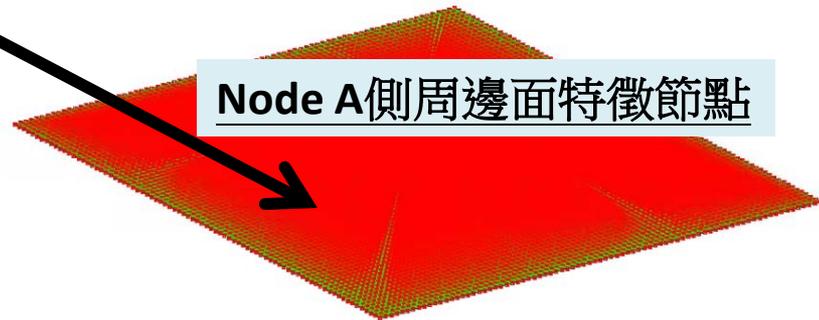
元素類型選取Rigid Link



自訂網格集Rigid Link1



切換使用面特徵選取節點

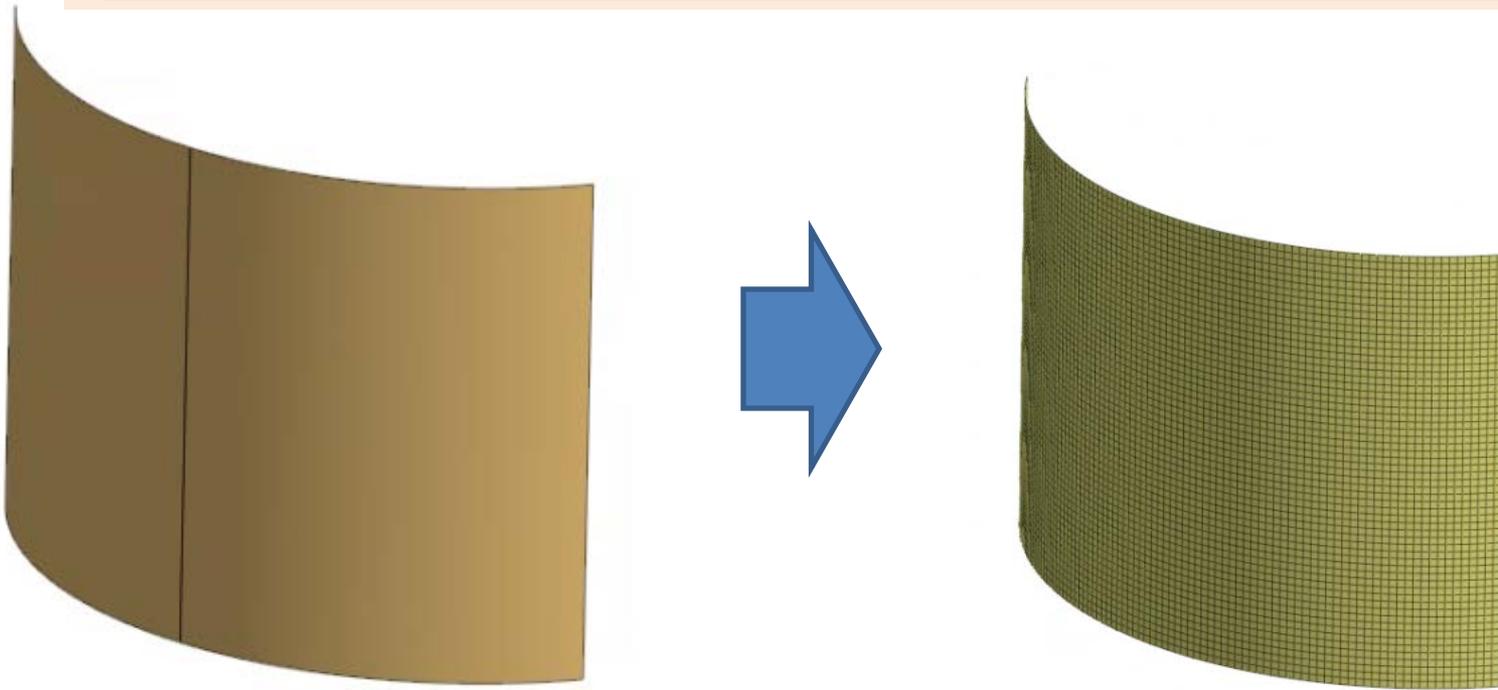


Node A側周邊面特徵節點

2D網格-鋼板



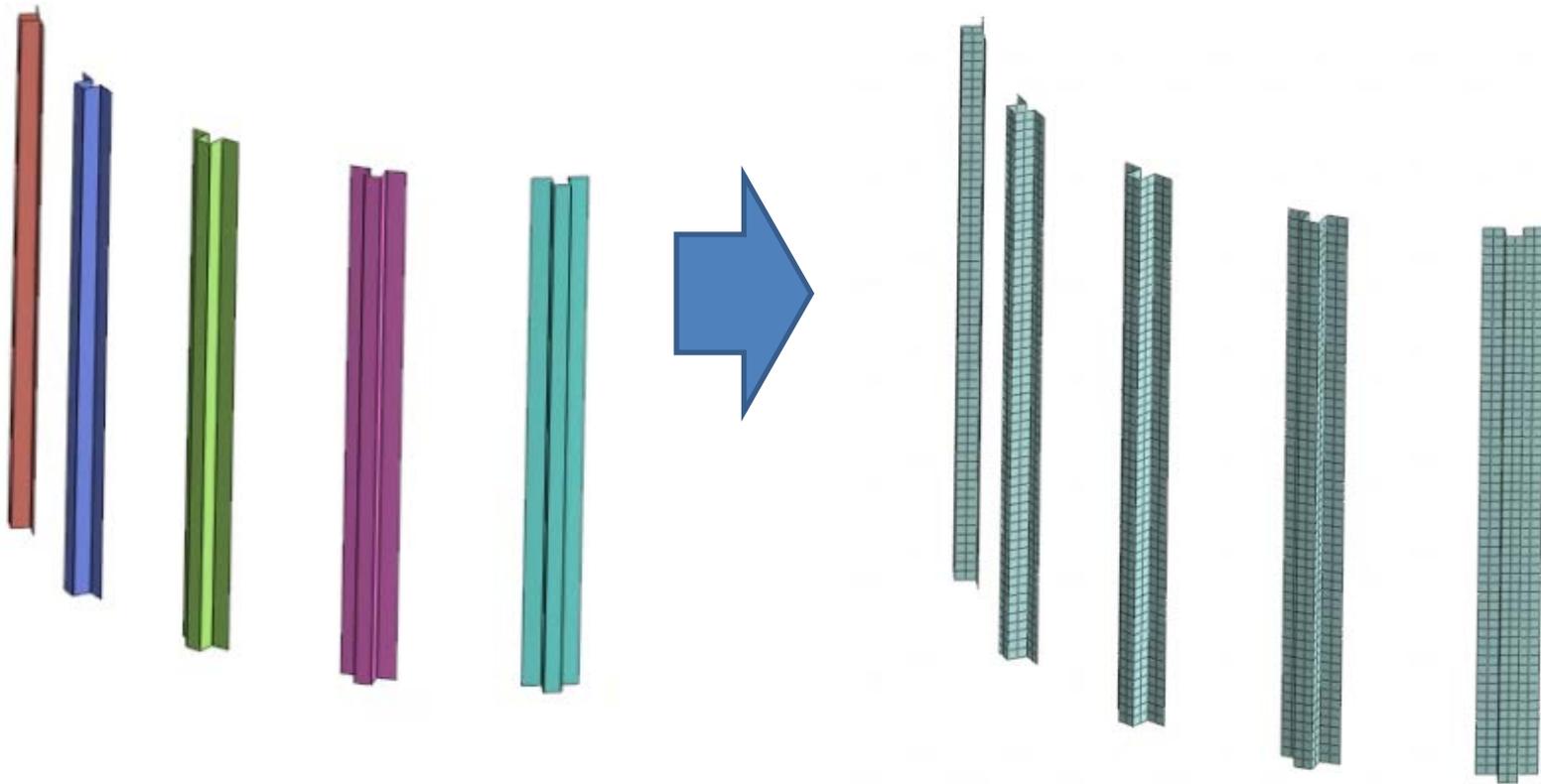
網格尺寸0.05(m)/屬性T0.025/自訂網格集名稱(鋼板)/合併節點/同一網格集



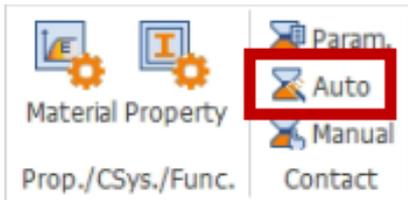
2D網格-立柱



網格尺寸0.05 (m)/屬性T0.012/自訂網格集名稱(立柱)/合併節點/同一網格集



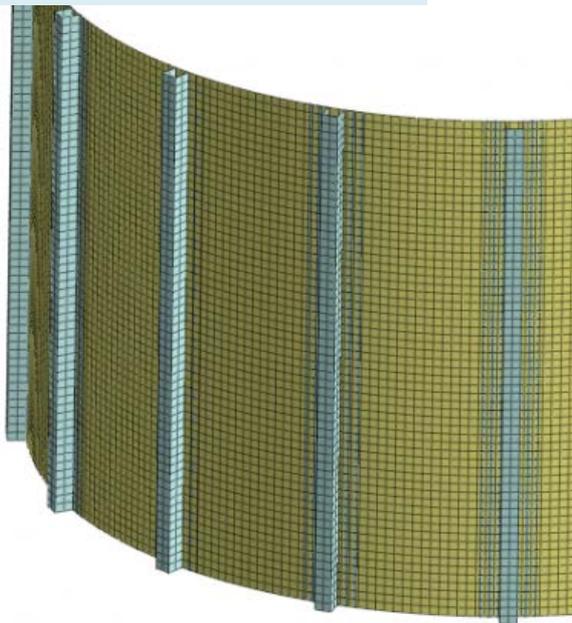
自動焊接接觸



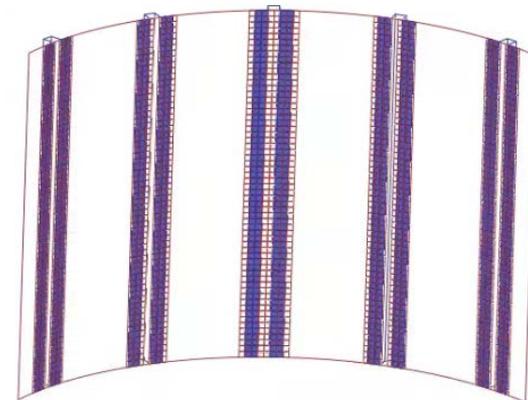
選取鋼板&立柱網格集



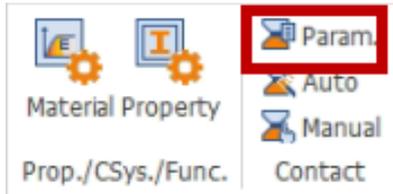
輸入判斷接觸間距0.01(m)



接觸檢視



接觸參數



Contact Parameters

ID: 4 Name: contact

Structural

Normal Stiffness Scaling Factor: 0.8

Tangential Stiffness Scaling: 0.01

Contact Tolerance: 0.001 m

Master Segment Extension Ratio: 0.005

Friction Coefficient:

Remove Initial Penetration by Adjusting Slave Nodes

Conduction for Seepage Flow: 0 m/sec/m

Heat Transfer Analysis

Thermal Conductance: 1000000 W/(m²*[T])

Breaking-Weld

Normal Failure Force: 0 N

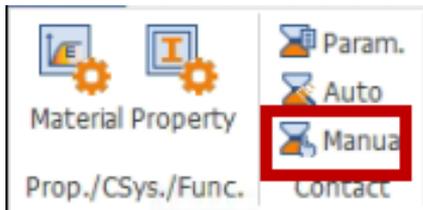
Shear Failure Force: 0 N

OK Cancel Apply

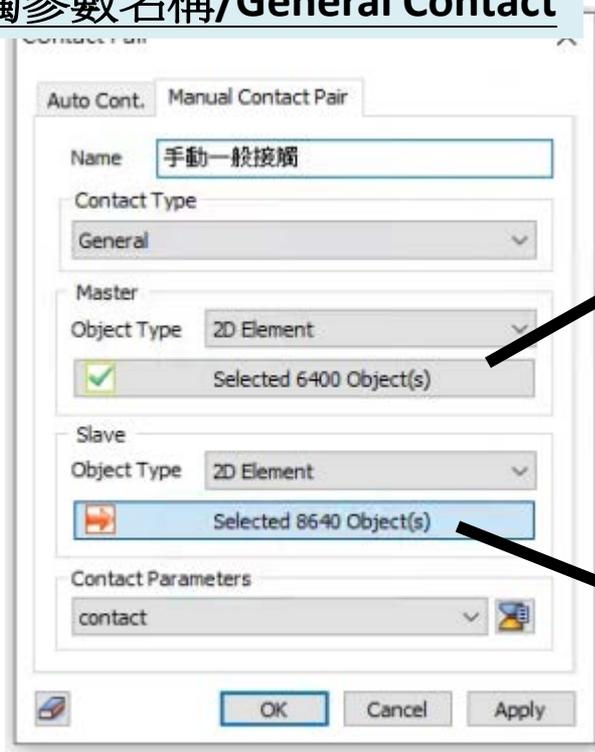
自定接觸參數名稱

調整接觸剛度/定義接觸間隙

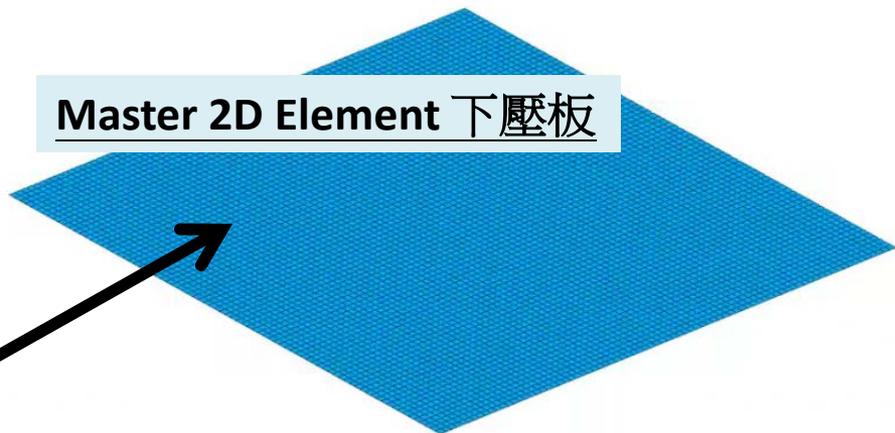
手動一般接觸



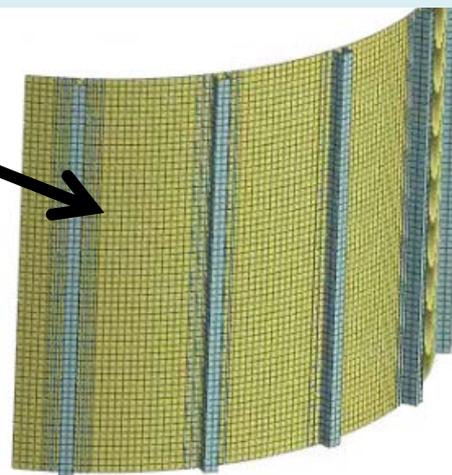
自定接觸參數名稱/General Contact



Master 2D Element 下壓板

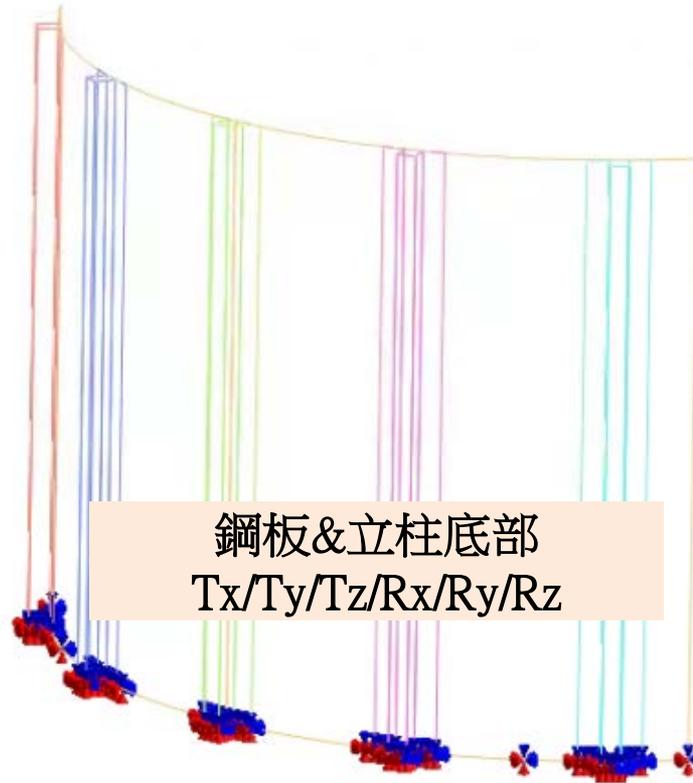


Slave 2D Element 鋼板&立柱

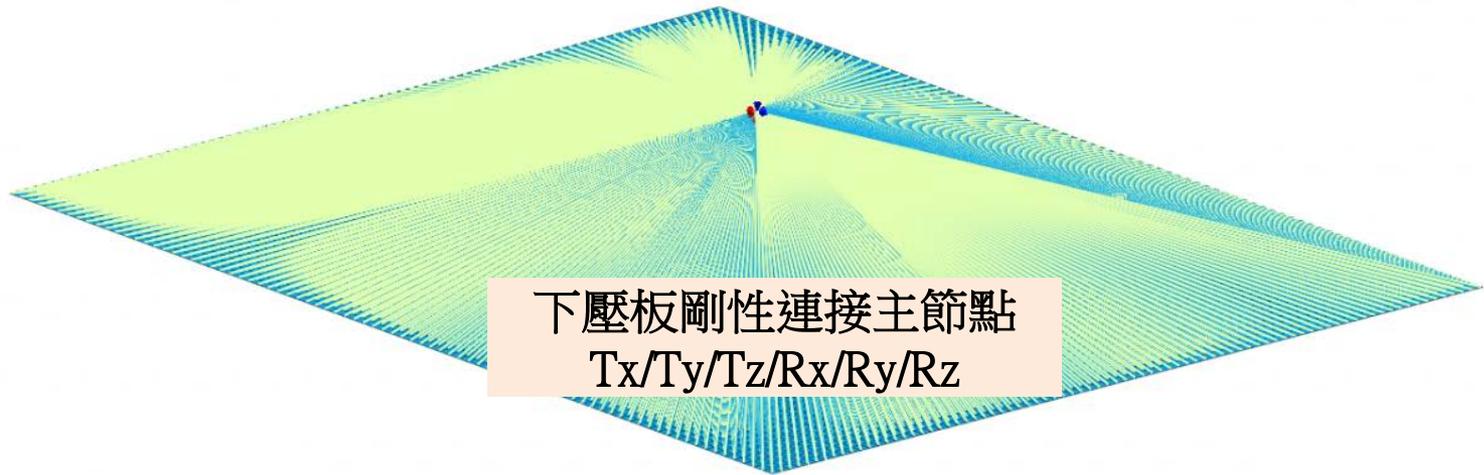


使用上一步驟接觸參數

邊界(鋼板&立柱)



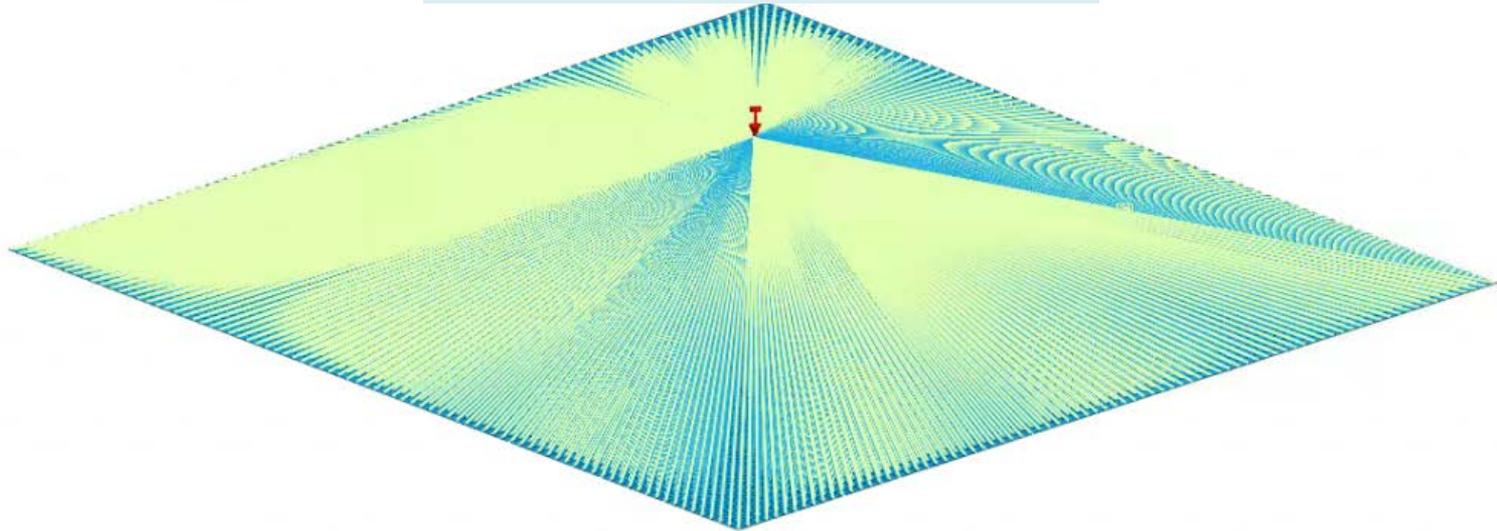
邊界(下壓板剛連接)



位移



依照Global座標施加 $T_z = -0.2(m)$



非線性分析-1

Id/Modify Analysis Case

Analysis Case Setting

Title: 後挫曲分析 **自訂析名稱**

Description:

Solution Type: Nonlinear Static

Analysis Control: [Icon]

Output Control: [Icon]

Construction Stage Set:

Analysis Case Model **分析類型: Nonlinear Static**

Mesh

- Default Mesh Set
- Manual Node A
- Rigid Link-1
- 下壓板
- 立柱
- 鋼板

Boundary Condition

- Boundary Set-1

Static Load

- Displacement

Contact Pair

- Auto-1-1
- 手動一般接觸

考慮所有網格集/邊界集/載荷集/接觸對

Solve Each Load Set Independently

Sorting: Name

OK Cancel Apply

非線性分析-2

非線性控制

預設計算材料非線性,勾選才計算幾何&邊界(接觸)非線性

增量數分成50等份/輸出每次增量結果

每一增量增加變形量 $0.2(m)/50=0.004(m)$

收斂判斷採Displacement(放寬收斂標準)

使用修正牛頓法

調整增量收斂計算次數

Analysis Control

General Nonlinear

Geometry Nonlinearity

Consider Geometric Nonlinear Effects

Update Pore Pressure with Deformation

Basic Nonlinear Parameters

Load Steps

Number of Increments: 50

Intermediate Output Request: Every Increment

Manual with User-Defined Steps: Load Step...

Time Steps

Time (Duration): 86400 sec

Number of Increments: 1

Iterative Scheme

General Enhanced Init Stress

Convergence Criteria / Error Tolerance

Displacement(U): 0.005

Load(P): 0.001

Work(W): 1e-006

Use Iteration Method

Arc-Length Method Displacement-Control Method

Min. Arc-Length Adjustment Ratio: 0.25

Max. Arc-Length Adjustment Ratio: 4

Max Arc-Length Increments: 5

Advanced Nonlinear Setting...

OK Cancel

Advanced Nonlinear Parameter

Nonlinear Solver Parameters

Use Default Settings

Stiffness Update Scheme Parameter

Stiffness Update Scheme: Modified Newton

Custom Update Method: Semi-Automatic(5)

No. of iterations before Stiffness Update (ITER and SEMI Methods): 50

Max. No. of Quasi-Newton Vectors: 5

Enhanced Predictor Disp.

Analysis Option

Terminate Analysis on Failed Convergence

Max No. of Iterations per Increment: 25

Max. Bisection Level: 10

Enable Line Search

Max. Line Search per Iteration: 4

Line Search: 5

Over-Relaxation: 1.2

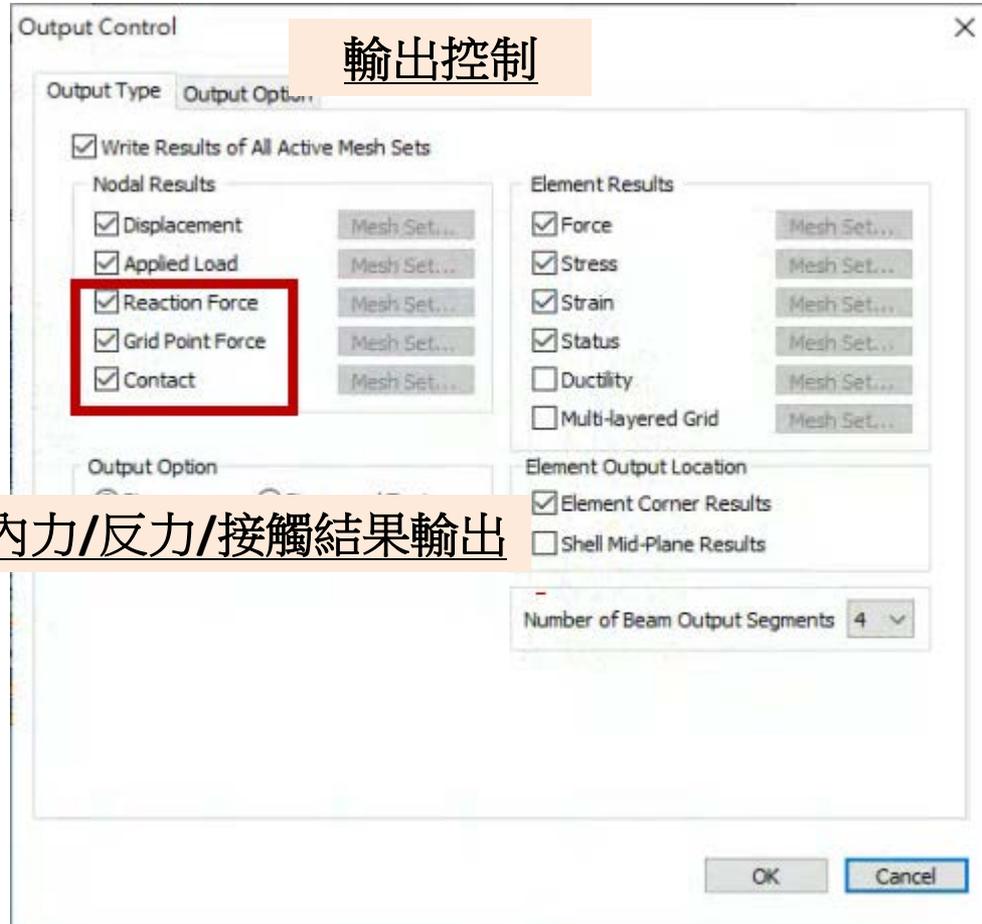
None

Divergence Threshold: 3

OK Cancel

註:後挫曲計算需考慮幾何非線性。

非線性分析-3



輸出控制

內力/反力/接觸結果輸出

執行分析

FEA NX Solver

Please wait! FEA NX Solver

Stop Execution

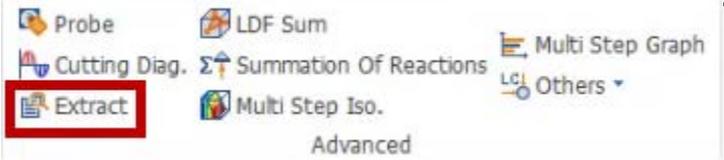
計算提示訊息(收斂迭代過程)

```
MAXIMUM STRESS : 0.0000E+00(1D:0), 3.0004E+07(2D:3221), 0.0000E+00(3D:0)
INCREMENT= 10 ( 20.00%), ITERATION= 8, ERROR NORMS: U( 1.63E-04/ 5.0E-03) CONVERGED
- RESULT SUMMARY
MAXIMUM TRANSLATION : -4.0000E-02(T3:1), MAXIMUM ROTATION : -4.6053E-02(R3:6595)
MAXIMUM STRESS : 0.0000E+00(1D:0), 2.8028E+08(2D:12389), 0.0000E+00(3D:0)
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 6718.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 7720.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 10647.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 11886.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 13638.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 9077.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 10765.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 6718.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Z OF NODE 8959.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 8133.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 6894.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Z OF NODE 12063.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Y OF NODE 11827.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-Z OF NODE 6718.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 7012.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 13335.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 9077.
WARNING [4005] : SINGULARITY IN SYSTEM MATRIX FIXED AT TRANSLATION-X OF NODE 9254.
NUMERICAL INSTABILITY DETECTED DURING ELEMENT COMPUTATIONS. BISECTING LOAD INCREMENT (BISECT LEVEL=1)
MAXIMUM NUMBER OF ITERATIONS REACHED. BISECTING LOAD INCREMENT (BISECT LEVEL=2)
MAXIMUM NUMBER OF ITERATIONS REACHED. BISECTING LOAD INCREMENT (BISECT LEVEL=3)
MAXIMUM NUMBER OF ITERATIONS REACHED. BISECTING LOAD INCREMENT (BISECT LEVEL=4)
MAXIMUM NUMBER OF ITERATIONS REACHED. BISECTING LOAD INCREMENT (BISECT LEVEL=5)

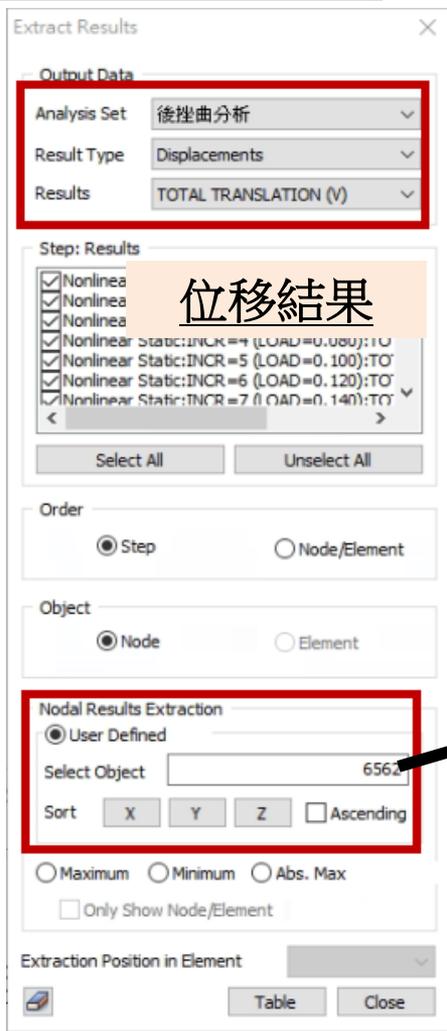
INCREMENT= 11 ( 20.06%), ITERATION= 4, ERROR NORMS: U( 1.55E-03/ 5.0E-03) CONVERGED
INCREMENT= 12 ( 20.13%), ITERATION= 6, ERROR NORMS: U( 3.32E-05/ 5.0E-03) CONVERGED
```

註:後挫曲屬於大變形計算,分析過程計算不穩定。

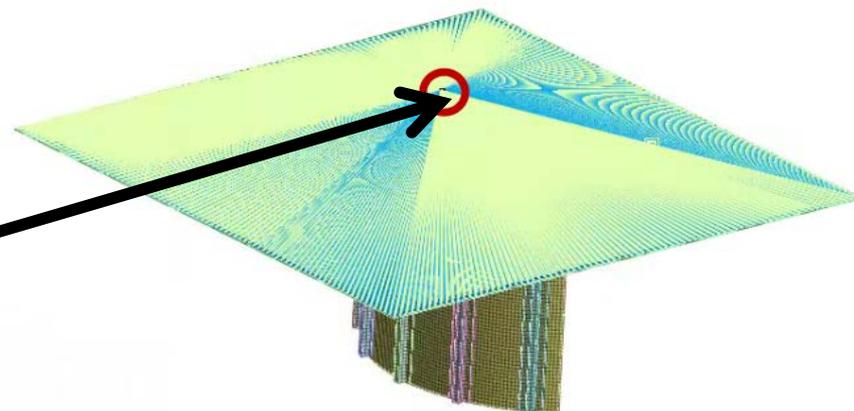
分析結果-1



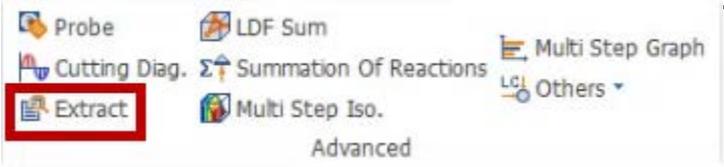
迭代過程剛性中心點位移結果



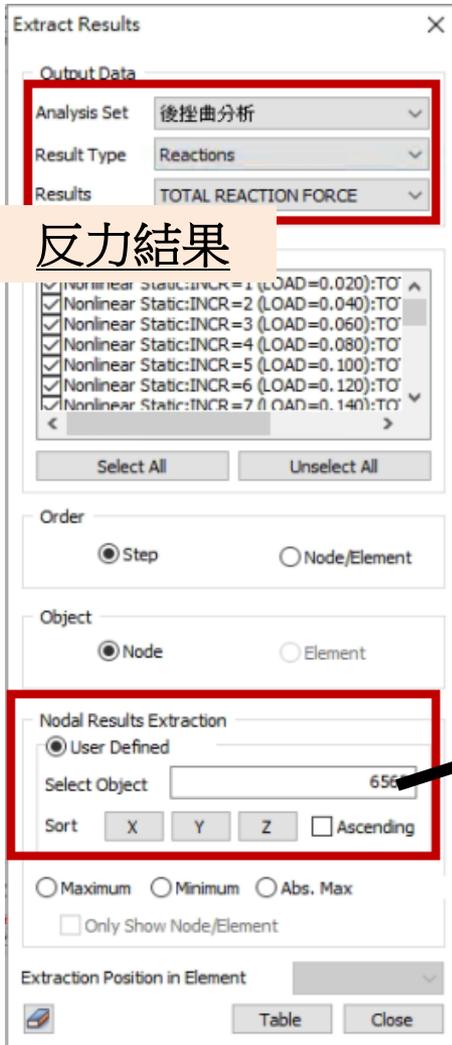
3	Nonlinear Static:INCR=3 (LOAD=0.060)	6.000000e-002	1.200000e-002
4	Nonlinear Static:INCR=4 (LOAD=0.080)	8.000000e-002	1.600000e-002
5	Nonlinear Static:INCR=5 (LOAD=0.100)	1.000000e-001	2.000000e-002
6	Nonlinear Static:INCR=6 (LOAD=0.120)	1.200000e-001	2.400000e-002
7	Nonlinear Static:INCR=7 (LOAD=0.140)	1.400000e-001	2.800000e-002
8	Nonlinear Static:INCR=8 (LOAD=0.160)	1.600000e-001	3.200000e-002
9	Nonlinear Static:INCR=9 (LOAD=0.180)	1.800000e-001	3.600000e-002
10	Nonlinear Static:INCR=10 (LOAD=0.200)	2.000000e-001	4.000000e-002
11	Nonlinear Static:INCR=16 (LOAD=0.220)	2.200000e-001	4.400000e-002
12	Nonlinear Static:INCR=17 (LOAD=0.240)	2.400000e-001	4.800000e-002
13	Nonlinear Static:INCR=19 (LOAD=0.260)	2.600000e-001	5.200000e-002
14	Nonlinear Static:INCR=20 (LOAD=0.280)	2.800000e-001	5.600000e-002
15	Nonlinear Static:INCR=21 (LOAD=0.300)	3.000000e-001	6.000000e-002
16	Nonlinear Static:INCR=22 (LOAD=0.320)	3.200000e-001	6.400000e-002
17	Nonlinear Static:INCR=23 (LOAD=0.340)	3.400000e-001	6.800000e-002
18	Nonlinear Static:INCR=24 (LOAD=0.360)	3.600000e-001	7.200000e-002
19	Nonlinear Static:INCR=25 (LOAD=0.380)	3.800000e-001	7.600000e-002
20	Nonlinear Static:INCR=26 (LOAD=0.400)	4.000000e-001	8.000000e-002
21	Nonlinear Static:INCR=27 (LOAD=0.420)	4.200000e-001	8.400000e-002
22	Nonlinear Static:INCR=28 (LOAD=0.440)	4.400000e-001	8.800000e-002
23	Nonlinear Static:INCR=29 (LOAD=0.460)	4.600000e-001	9.200000e-002
24	Nonlinear Static:INCR=30 (LOAD=0.480)	4.800000e-001	9.600000e-002
25	Nonlinear Static:INCR=31 (LOAD=0.500)	5.000000e-001	1.000000e-001
26	Nonlinear Static:INCR=32 (LOAD=0.520)	5.200000e-001	1.040000e-001
27	Nonlinear Static:INCR=33 (LOAD=0.540)	5.400000e-001	1.080000e-001
28	Nonlinear Static:INCR=34 (LOAD=0.560)	5.600000e-001	1.120000e-001
29	Nonlinear Static:INCR=35 (LOAD=0.580)	5.800000e-001	1.160000e-001



分析結果-2



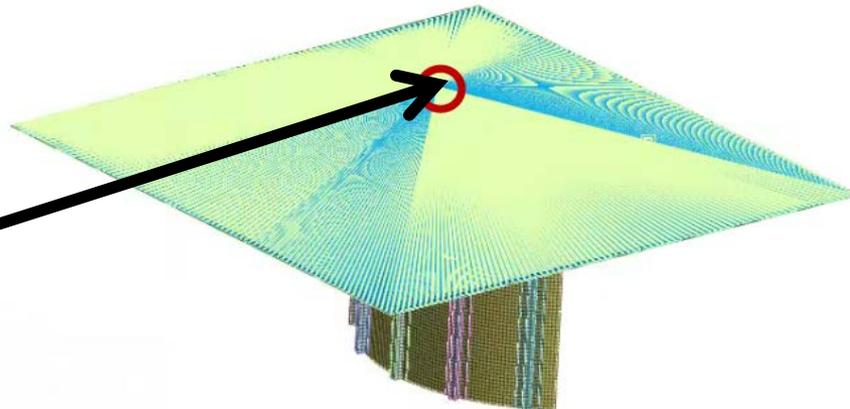
迭代過程剛性中心點反力結果



反力結果

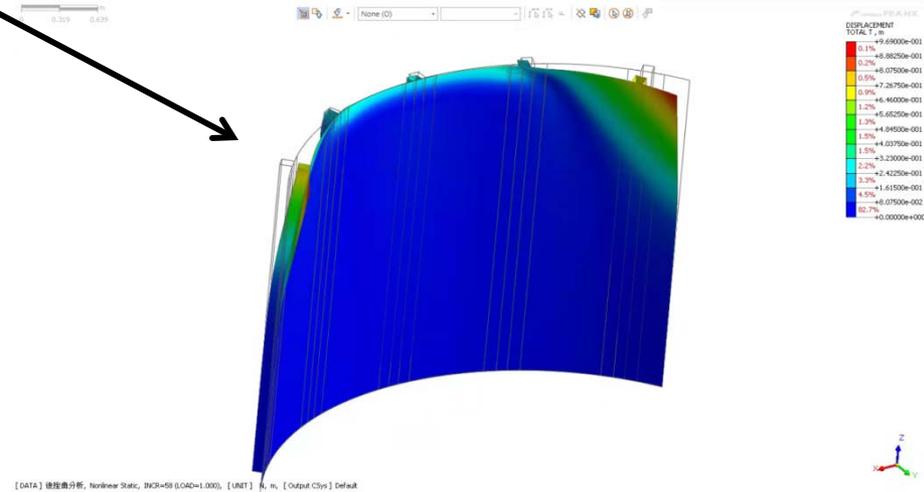
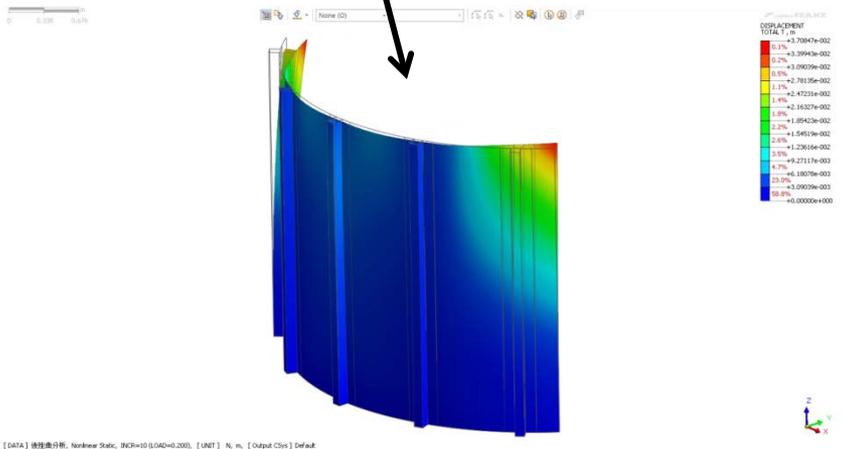
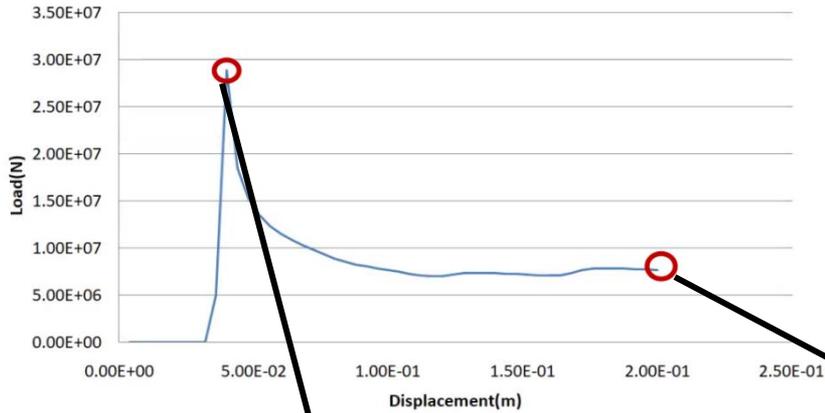
2	Nonlinear Static:INCR=2 (LOAD=0.040)	4.000000e-002	0.000000e+000
3	Nonlinear Static:INCR=3 (LOAD=0.060)	6.000000e-002	0.000000e+000
4	Nonlinear Static:INCR=4 (LOAD=0.080)	8.000000e-002	0.000000e+000
5	Nonlinear Static:INCR=5 (LOAD=0.100)	1.000000e-001	0.000000e+000
6	Nonlinear Static:INCR=6 (LOAD=0.120)	1.200000e-001	0.000000e+000
7	Nonlinear Static:INCR=7 (LOAD=0.140)	1.400000e-001	0.000000e+000
8	Nonlinear Static:INCR=8 (LOAD=0.160)	1.600000e-001	7.642497e+003
9	Nonlinear Static:INCR=9 (LOAD=0.180)	1.800000e-001	4.984495e+006
10	Nonlinear Static:INCR=10 (LOAD=0.200)	2.000000e-001	2.883290e+007
11	Nonlinear Static:INCR=16 (LOAD=0.220)	2.200000e-001	1.842622e+007
12	Nonlinear Static:INCR=17 (LOAD=0.240)	2.400000e-001	1.527795e+007
13	Nonlinear Static:INCR=19 (LOAD=0.260)	2.600000e-001	1.350124e+007
14	Nonlinear Static:INCR=20 (LOAD=0.280)	2.800000e-001	1.232806e+007
15	Nonlinear Static:INCR=21 (LOAD=0.300)	3.000000e-001	1.148340e+007
16	Nonlinear Static:INCR=22 (LOAD=0.320)	3.200000e-001	1.083479e+007
17	Nonlinear Static:INCR=23 (LOAD=0.340)	3.400000e-001	1.029085e+007
18	Nonlinear Static:INCR=24 (LOAD=0.360)	3.600000e-001	9.789326e+006
19	Nonlinear Static:INCR=25 (LOAD=0.380)	3.800000e-001	9.305513e+006
20	Nonlinear Static:INCR=26 (LOAD=0.400)	4.000000e-001	8.898454e+006
21	Nonlinear Static:INCR=27 (LOAD=0.420)	4.200000e-001	8.541397e+006
22	Nonlinear Static:INCR=28 (LOAD=0.440)	4.400000e-001	8.251520e+006
23	Nonlinear Static:INCR=29 (LOAD=0.460)	4.600000e-001	8.023455e+006
24	Nonlinear Static:INCR=30 (LOAD=0.480)	4.800000e-001	7.824082e+006
25	Nonlinear Static:INCR=31 (LOAD=0.500)	5.000000e-001	7.633784e+006
26	Nonlinear Static:INCR=32 (LOAD=0.520)	5.200000e-001	7.451970e+006
27	Nonlinear Static:INCR=33 (LOAD=0.540)	5.400000e-001	7.281841e+006
28	Nonlinear Static:INCR=34 (LOAD=0.560)	5.600000e-001	7.123520e+006
29	Nonlinear Static:INCR=35 (LOAD=0.580)	5.800000e-001	6.984035e+006
30	Nonlinear Static:INCR=36 (LOAD=0.600)	6.000000e-001	7.007681e+006

1



分析結果-3

28832900(N)=>近似線性挫曲結果



[DATA] 總挫曲分析, Nonlinear Static, BCR=10 (LOAD=0.200), [UNIT] N, m, [Output Csys] Default

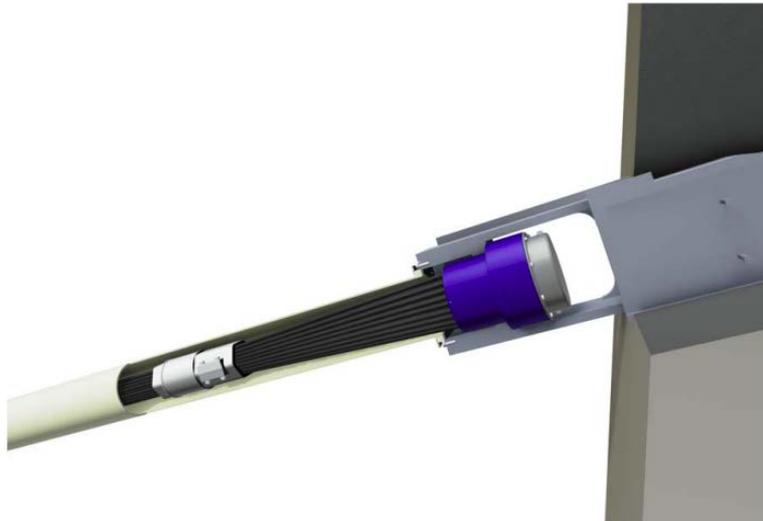
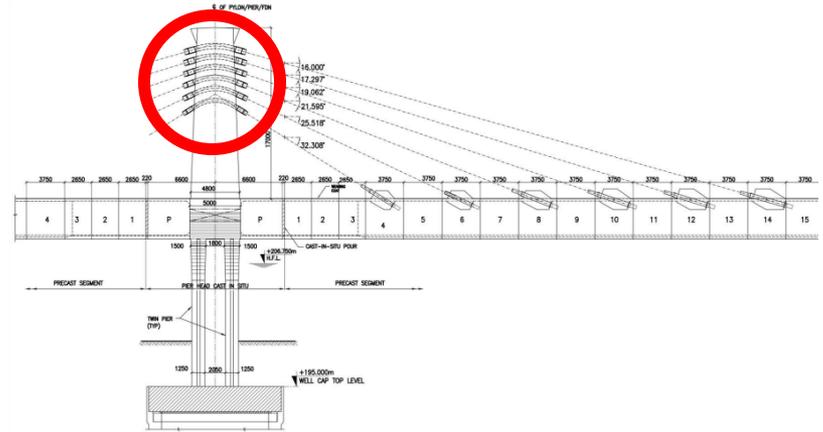
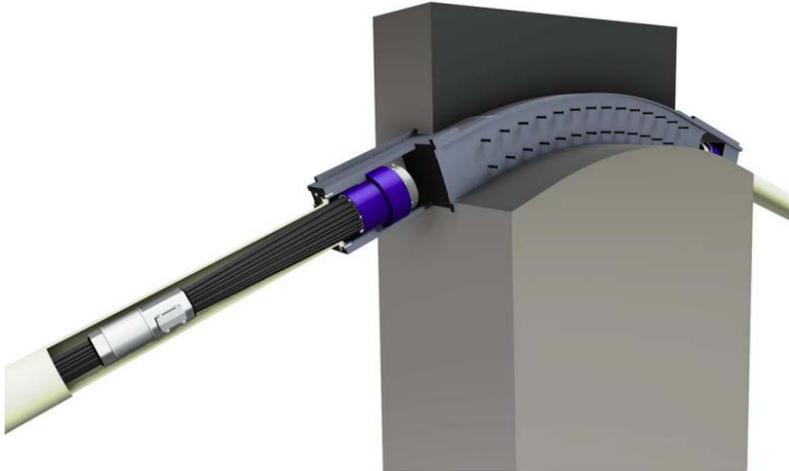
[DATA] 總挫曲分析, Nonlinear Static, BCR=50 (LOAD=1.000), [UNIT] N, m, [Output Csys] Default

FEA NX_標準教學系列

鋼骨結構-疲勞分析
鏈接錨箱系統

台灣邁達斯

鏈接錨箱系統 (Link Anchor Box System)



Reference

<https://dywidag.com/>

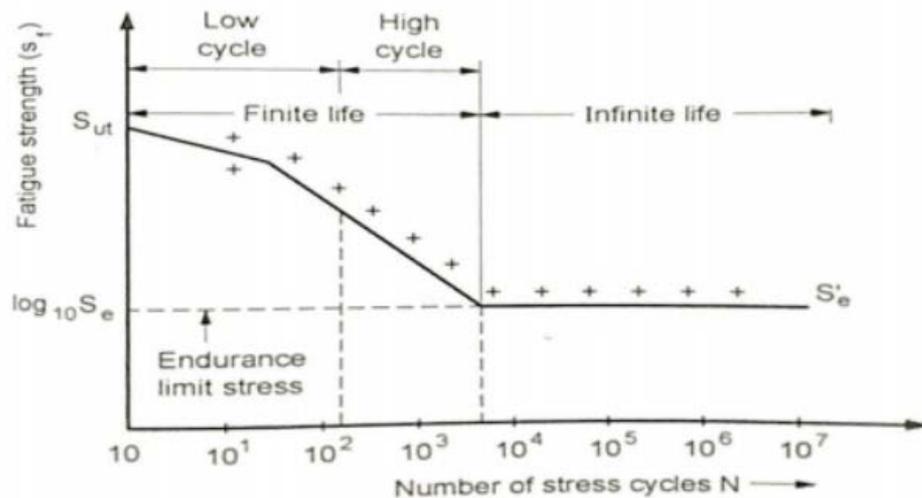


S-N Curve for Steel

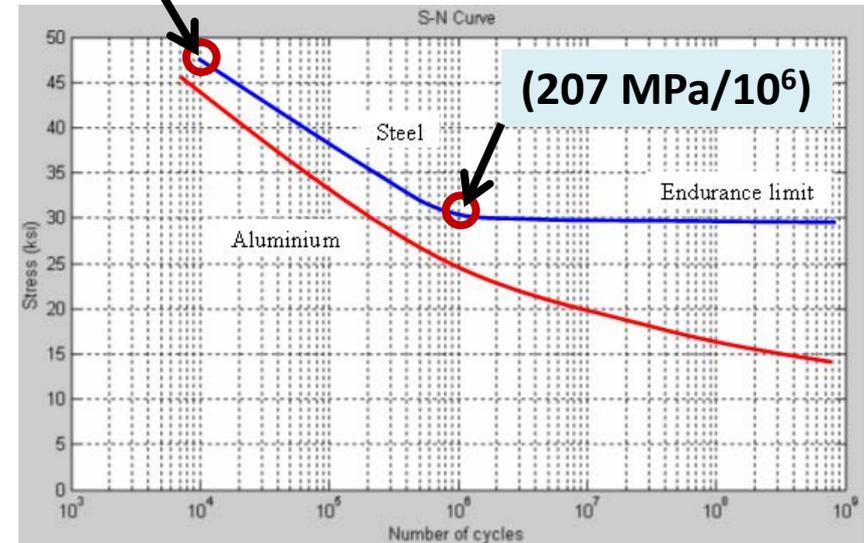
Endurance Limit

It is defined as maximum value of the completely reversed bending stress which a polished standard specimen can withstand without failure, for infinite number of cycles (usually 10^7 cycles).

S-N Curve for steel:



(327 MPa/ 10^4)



(207 MPa/ 10^6)

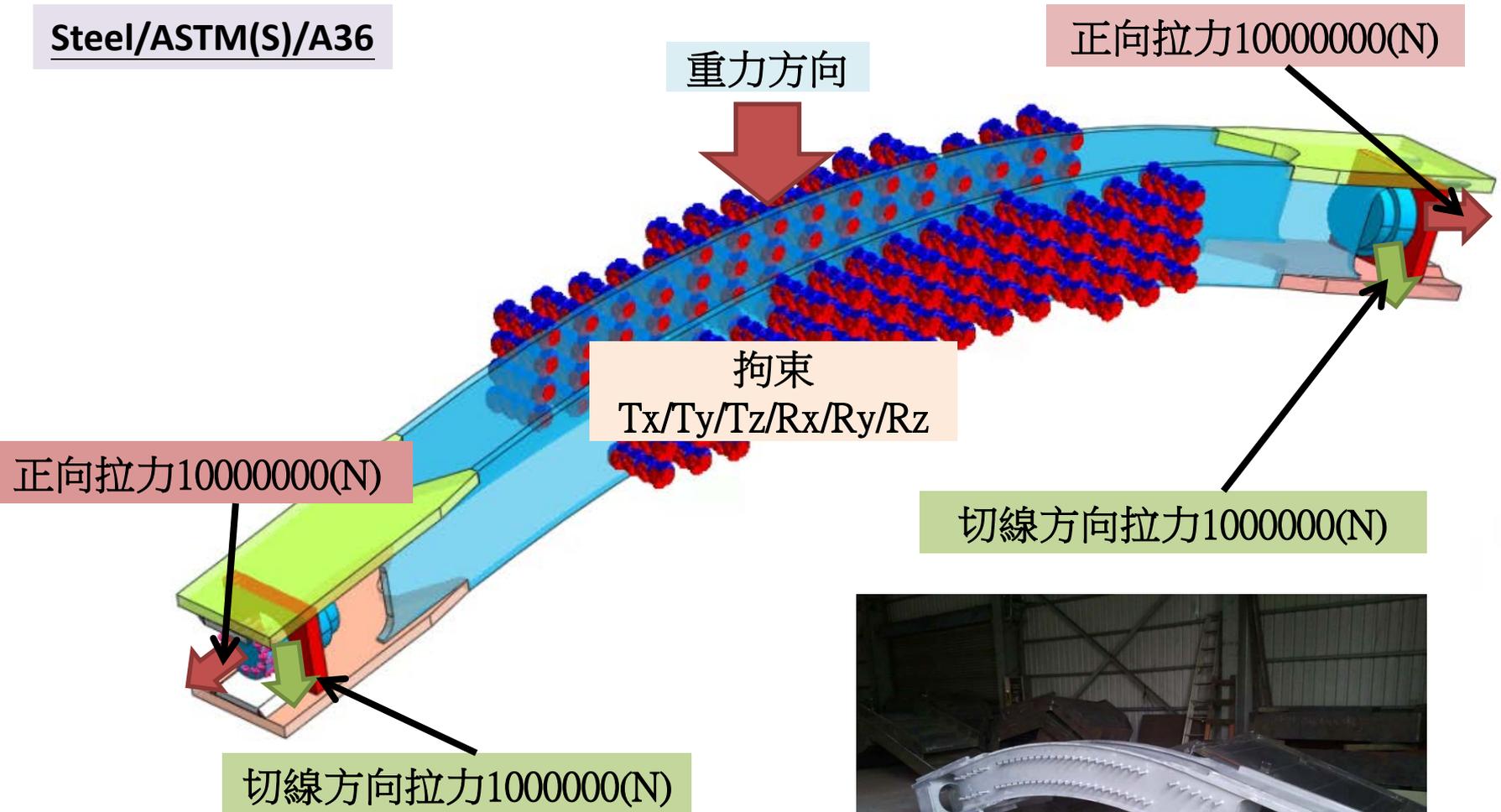
Reference

https://en.wikipedia.org/wiki/Fatigue_limit

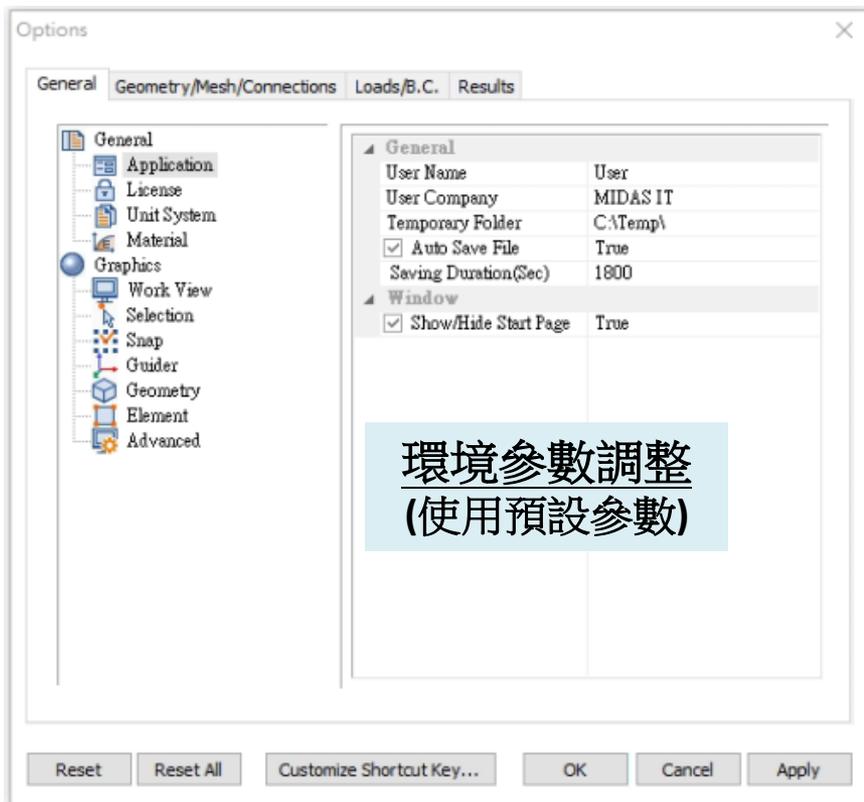
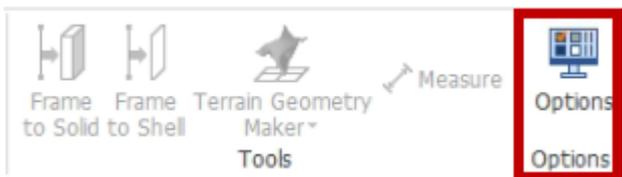
<https://mechdiploma.com/define-endurance-limit-and-draw-typical-s-n-curve-steel>

分析說明

Steel/ASTM(S)/A36



環境

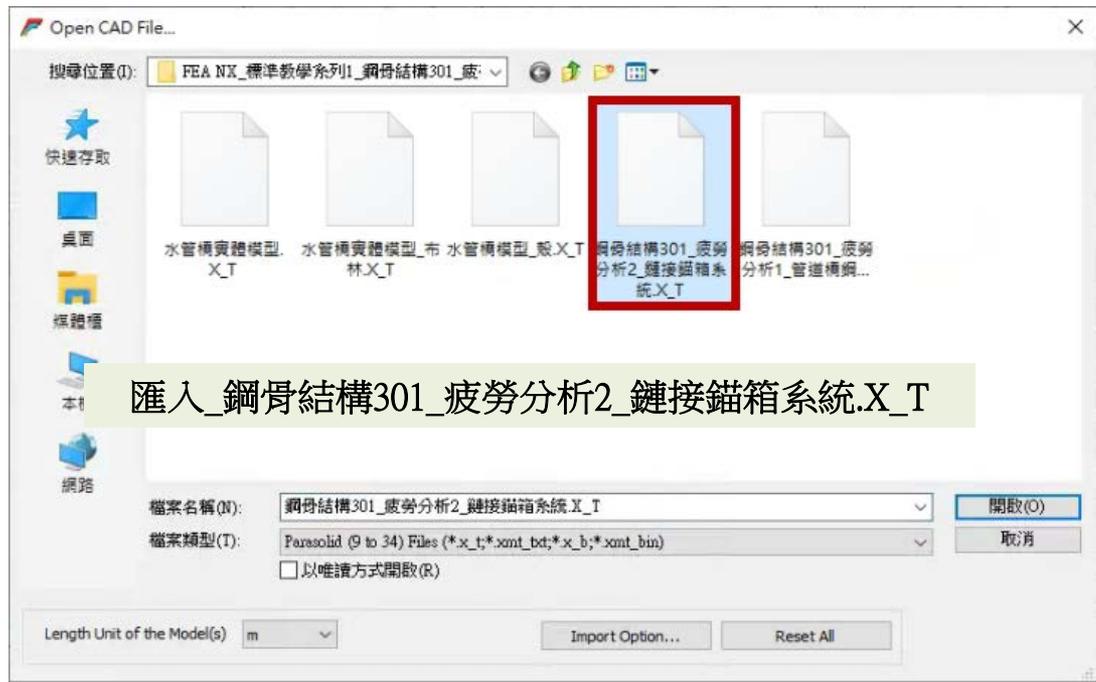
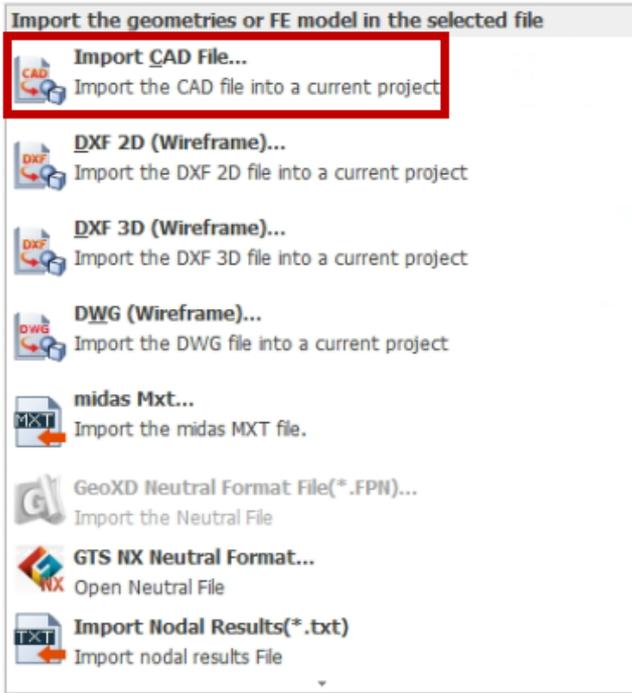
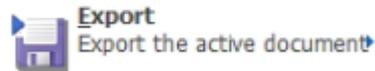


新文件

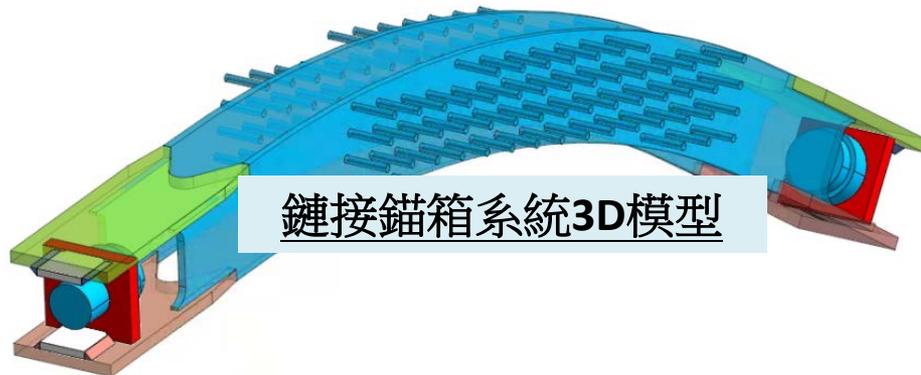


單位使用N/mm/J/sec

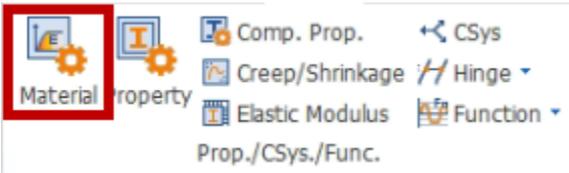
匯入模型



匯入_鋼骨結構301_疲勞分析2_鏈接錨箱系統.X_T



材料



自訂材料名稱A36

材料模型:Elastic

勾選Structure

Material ID: [dropdown]

Model Type: Elastic [dropdown] Structure

General | Thermal | Time Dependent | Temperature Dependent

Elastic Modulus(E): 199948.024 N/mm²

Inc. of Elastic Modulus: 0 N/mm²

Inc. of Elastic Modulus Ref. Height: 0 mm

Poisson's Ratio(ν): 0.3

Unit Weight(γ): 7.70910111e-0 N/mm³

Initial Stress Parameters

Ko Determination: 1

Automatic Manual Anisotropy

Thermal Parameter

Thermal Coefficient: 1.17e-005 1/[T]

Molecular vapor diffusion coefficient: 0 mm²/sec

Thermal diffusion enhancement: 0

Damping Ratio(For Dynamic)

Damping Ratio: 0.05

Safety Result(Mohr-Coulomb)

Cohesion(C): 0.03 N/mm²

Frictional Angle(Φ): 36 [deg]

Tensile Strength: 0 N/mm²

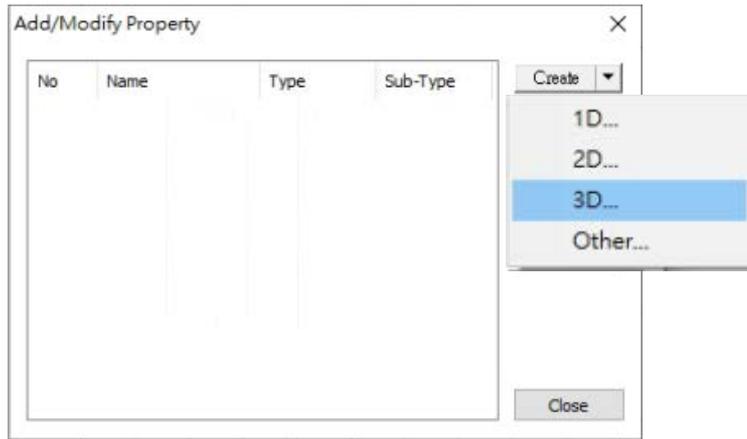
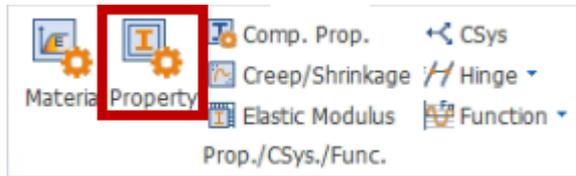
DB

OK Cancel Apply

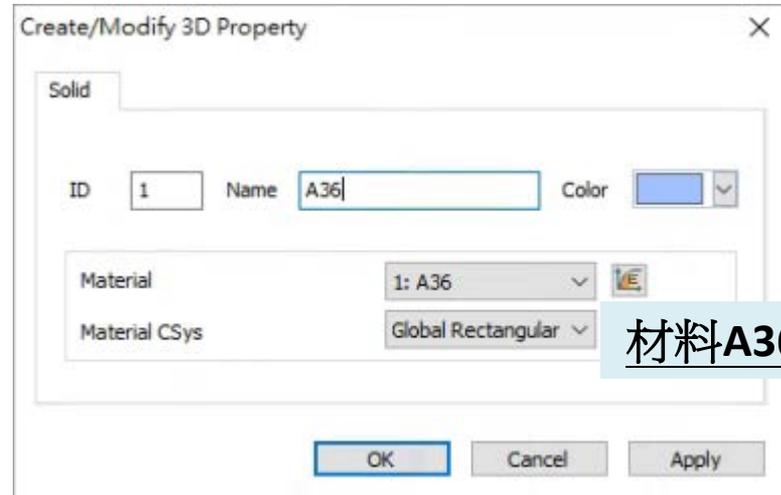
Steel/ASTM(S)/A36

註:範例相關參數使用假設條件。

屬性



新增A36



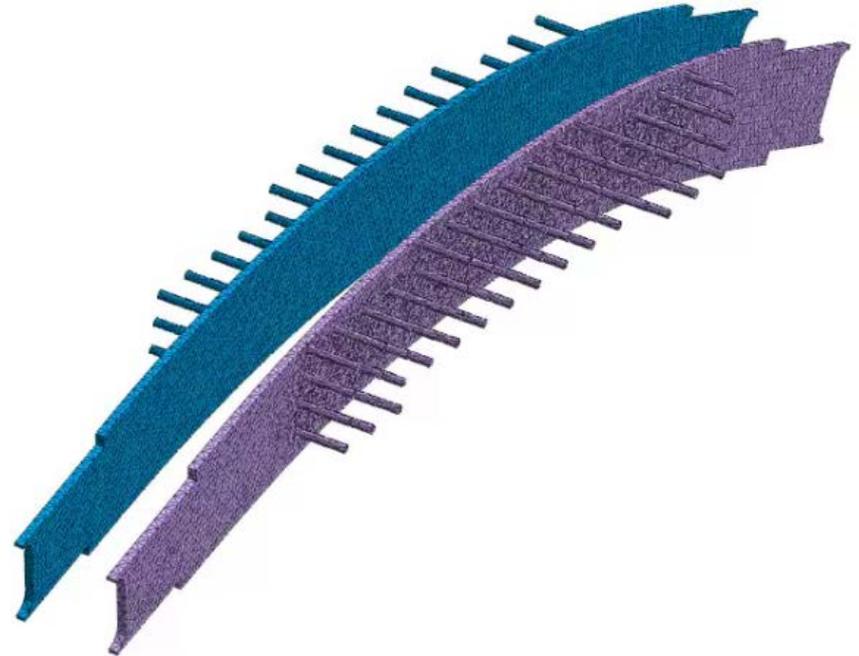
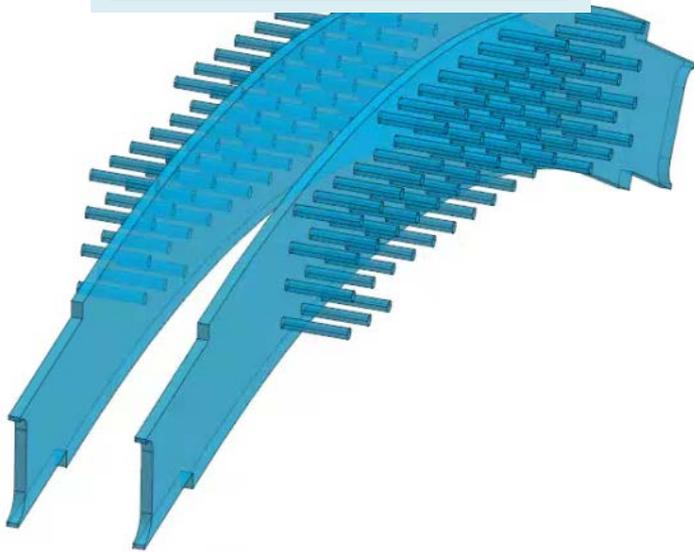
材料A36

3D網格-1



四面體一階/網格尺寸50(mm)/屬性A36 /不共節點/
自訂網格集名稱

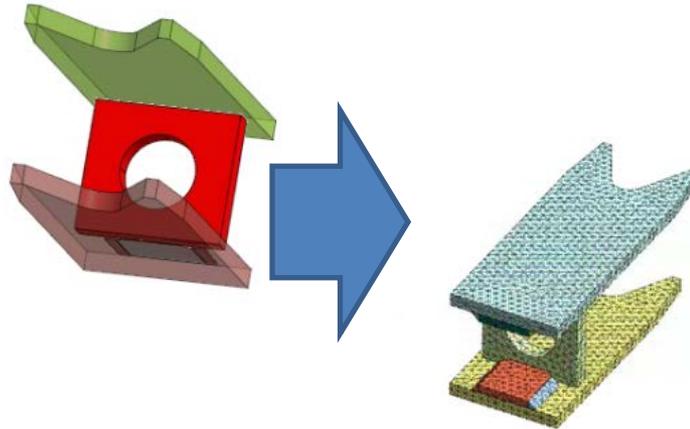
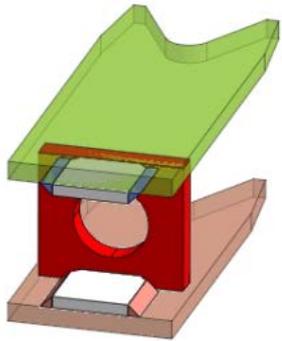
Web Plates in concrete



3D網格-2



四面體二階/網格尺寸50(mm)/屬性A36/不共節點/
自訂網格集名稱

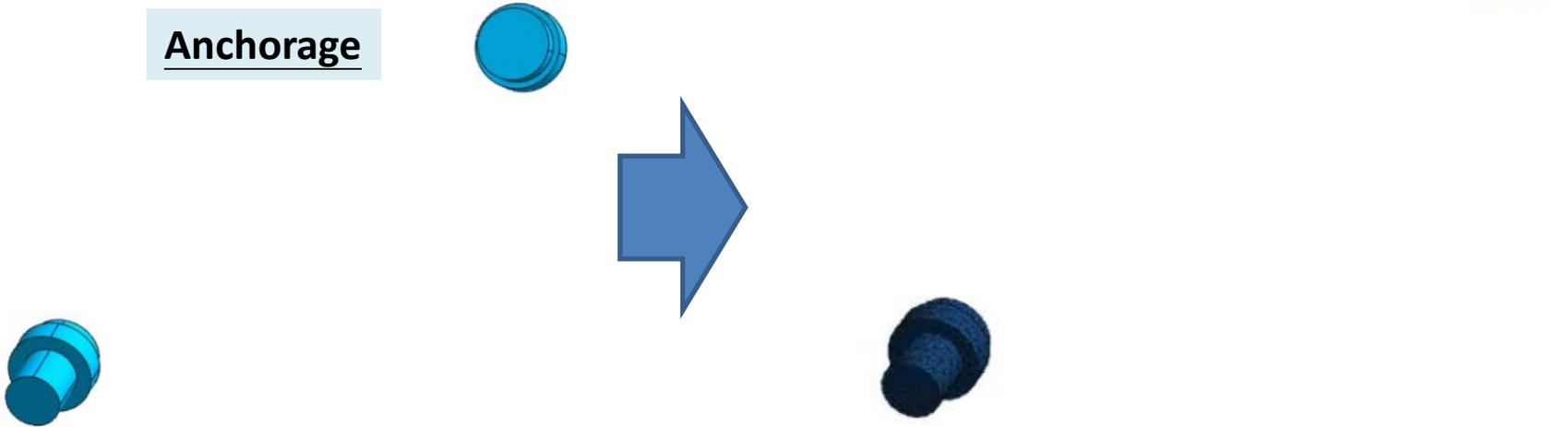


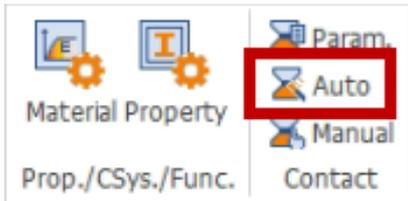
3D網格-3



四面體一階/網格尺寸50(mm)/屬性A36/不共節點/
自訂網格集名稱

Anchorage

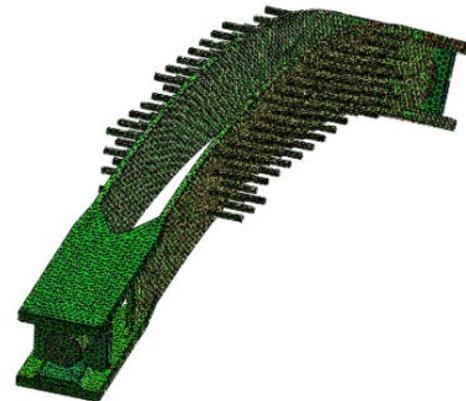
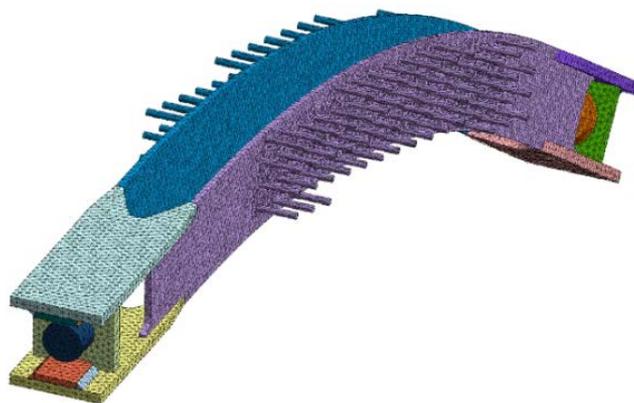




自動焊接接觸

顯示所有網格集

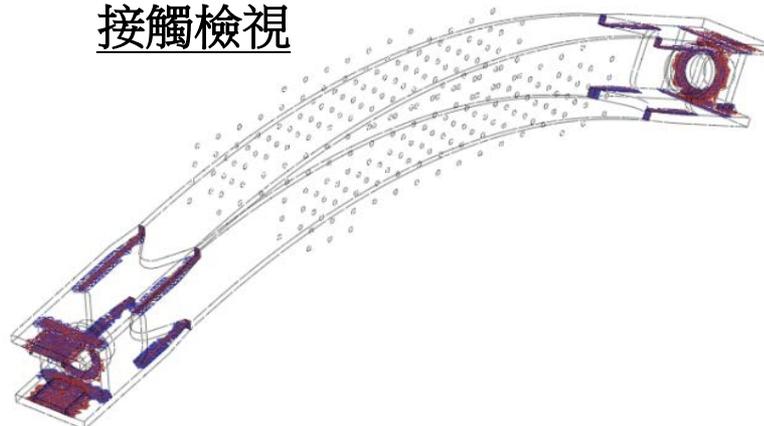
框選所有網格集



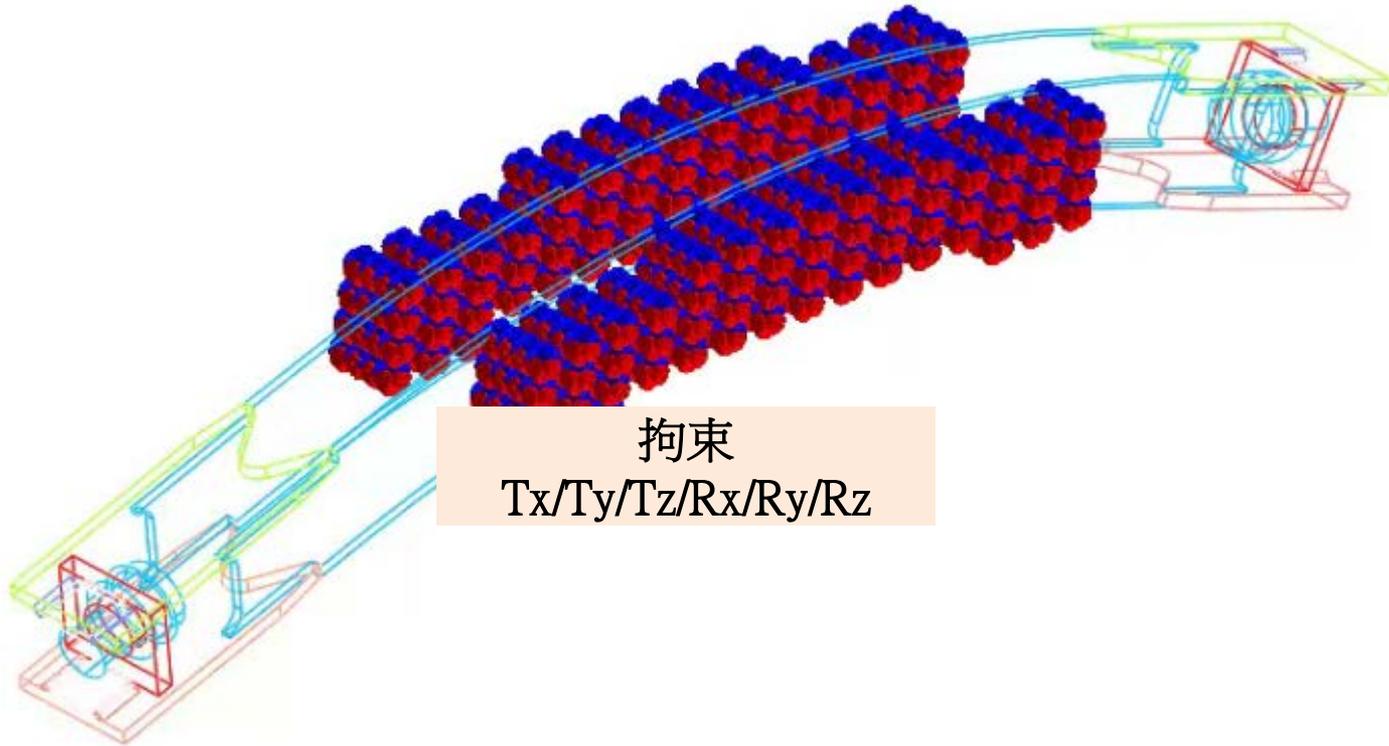
預設接觸類型:Welde

自動接觸間距

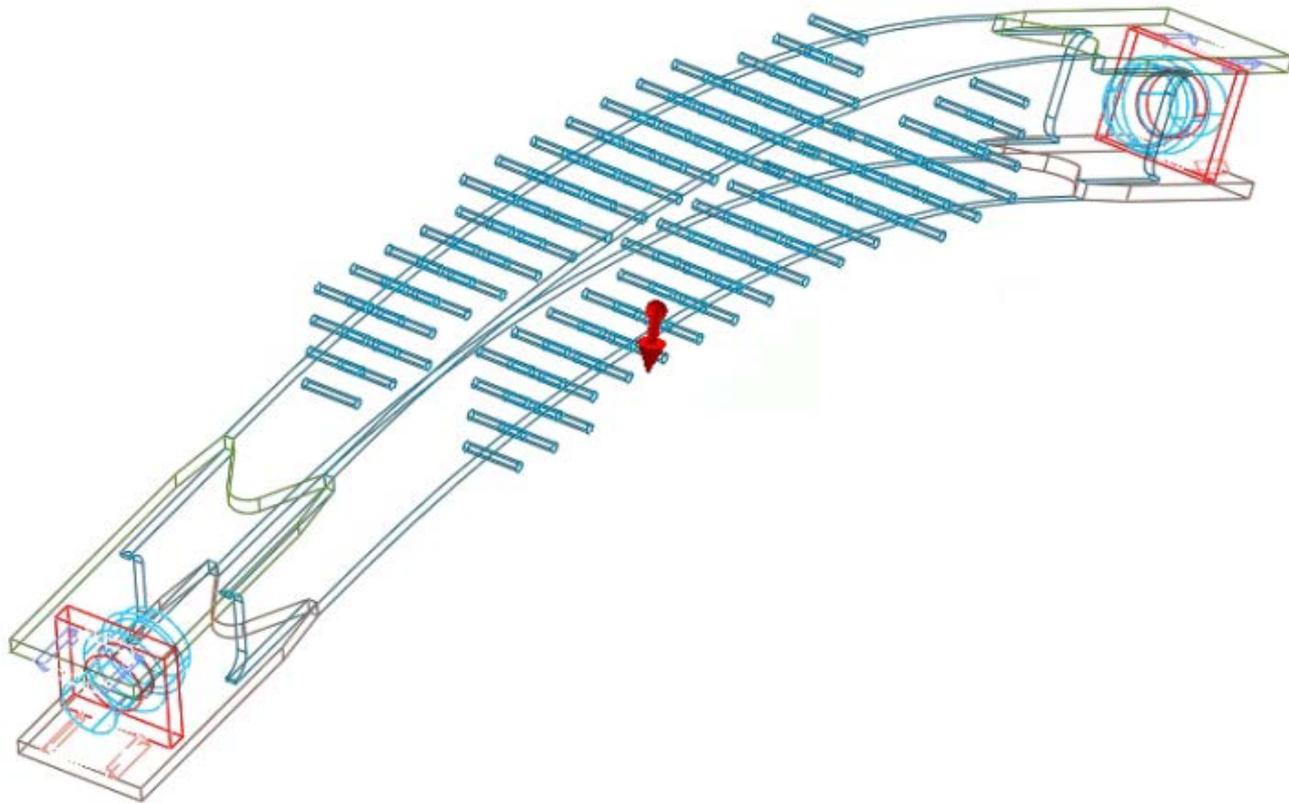
接觸檢視



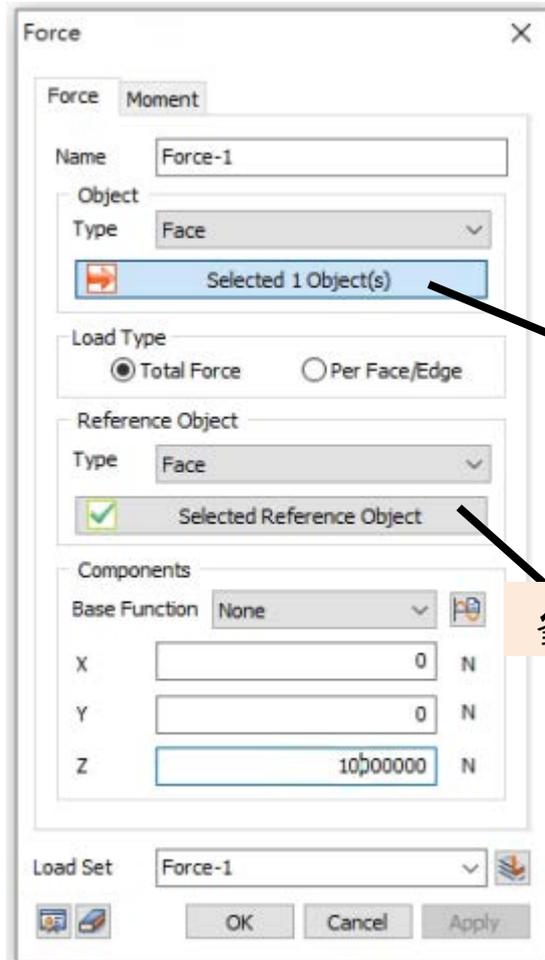
邊界



自重

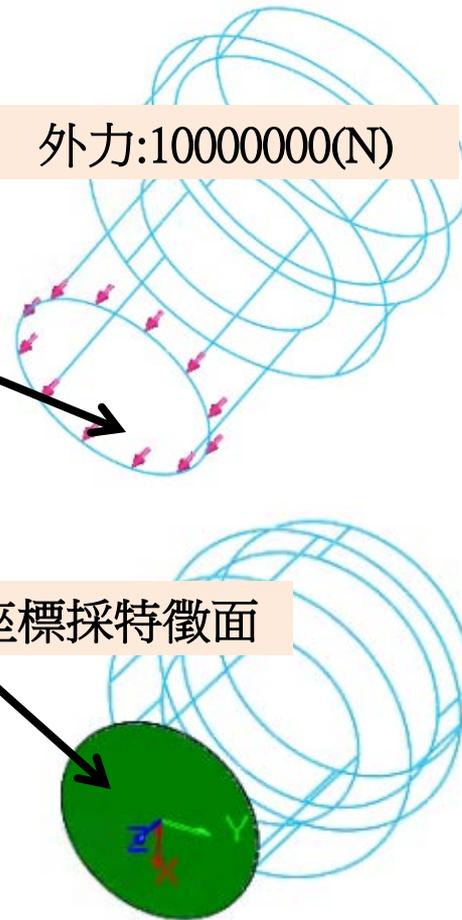


外力-1

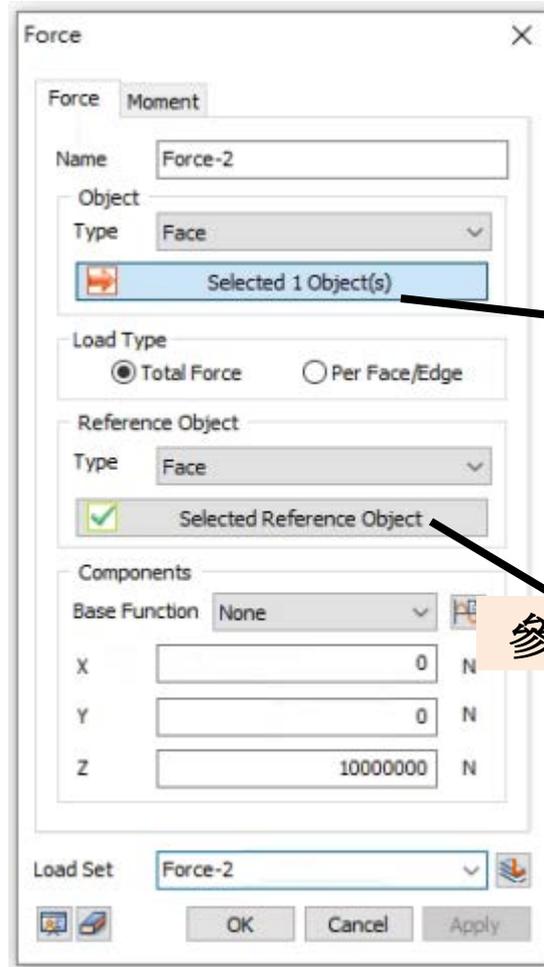


外力:10000000(N)

參考座標採特徵面

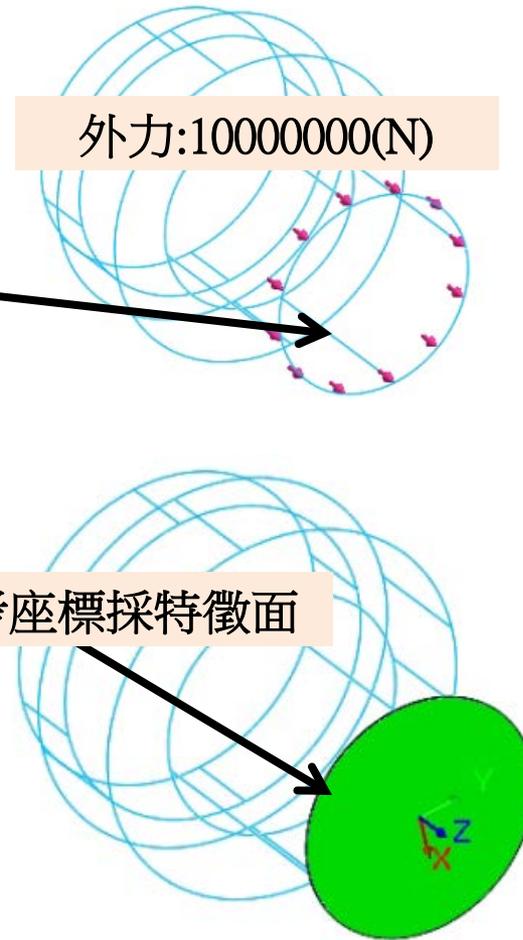


外力-2

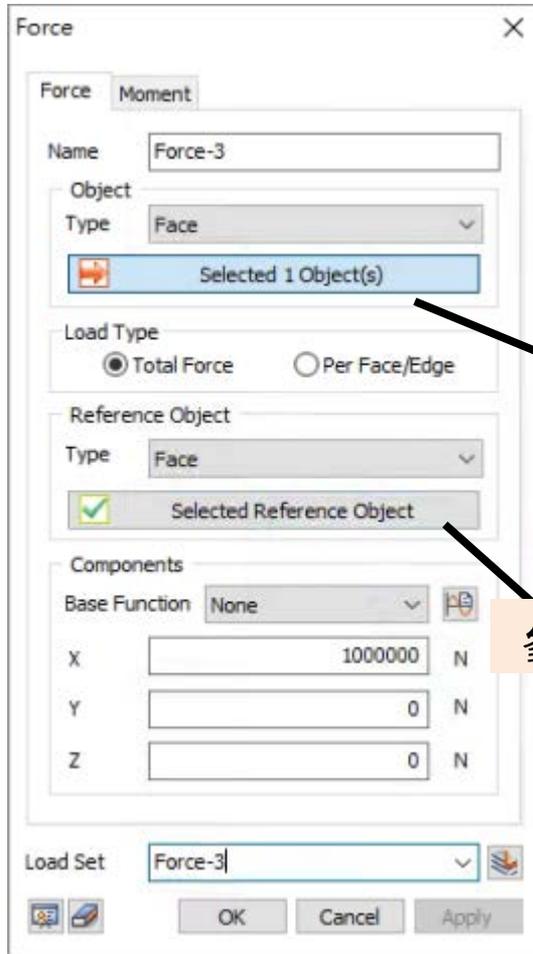


外力:10000000(N)

參考座標採特徵面

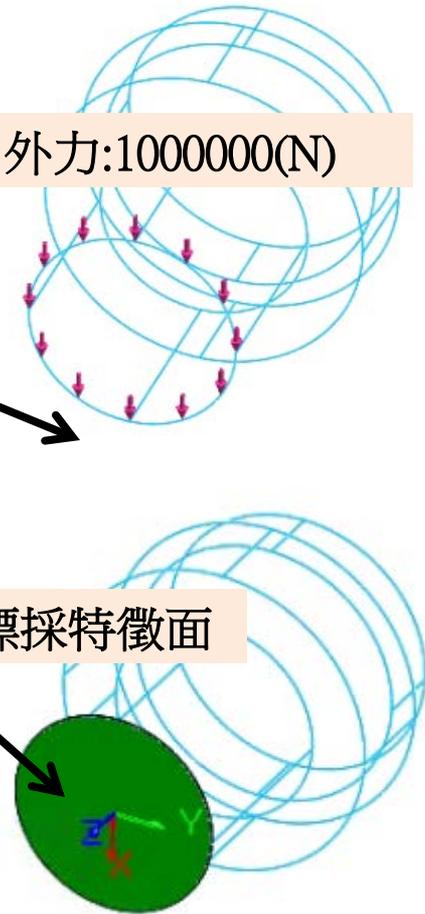


外力-3



外力:1000000(N)

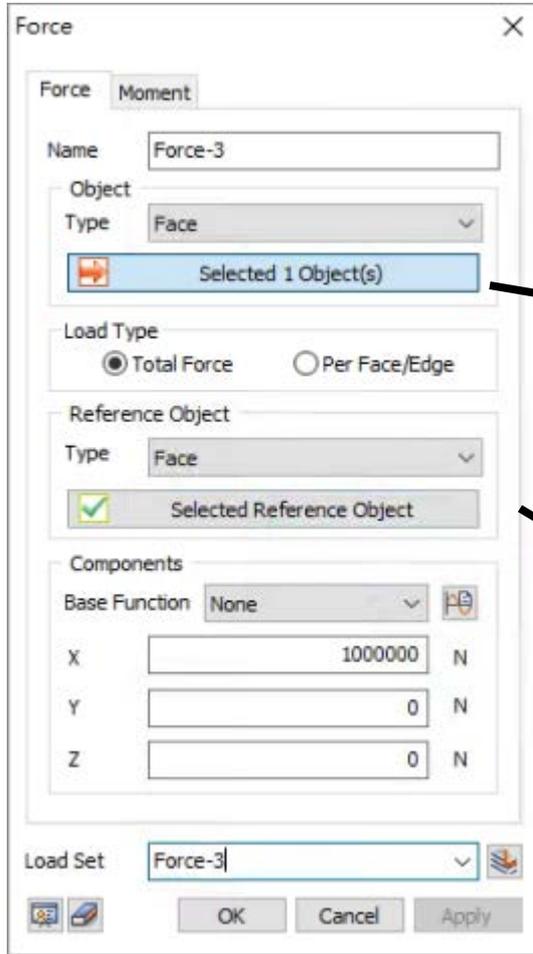
參考座標採特徵面



註:範例相關參數使用假設條件。

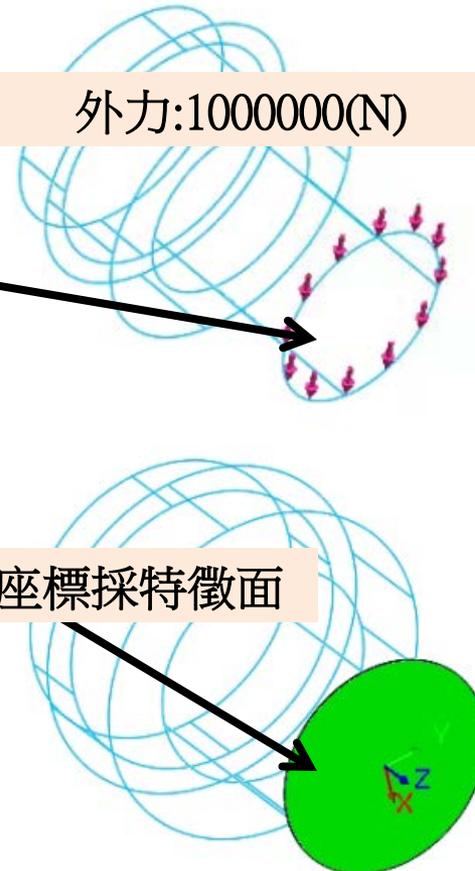


外力-4



外力:1000000(N)

參考座標採特徵面



註:範例相關參數使用假設條件。





線性分析

Add/Modify Analysis Case

Analysis Case Setting

Title: 管道橋網構件 **自訂分析名稱**

Description:

Solution Type: Linear Static

Analysis Control

Output Control

Construction Stage Set:

Analysis Case Model

All Sets << >> Active Sets

Mesh

- Auto-Mesh(3D)
- Auto-Mesh(3D)-1
- Auto-Mesh(3D)-10
- Auto-Mesh(3D)-11
- Auto-Mesh(3D)-12
- Auto-Mesh(3D)-13
- Auto-Mesh(3D)-17
- Auto-Mesh(3D)-18
- Auto-Mesh(3D)-19
- Auto-Mesh(3D)-2
- Auto-Mesh(3D)-3
- Auto-Mesh(3D)-4
- Auto-Mesh(3D)-5
- Auto-Mesh(3D)-6
- Auto-Mesh(3D)-7
- Auto-Mesh(3D)-8
- Auto-Mesh(3D)-9
- Default Mesh Set
- Web Plates in concrete

分析類型: Linear Static

考慮所有網格集/邊界集/載荷集/接觸對

Solve Each Load Set Independently

Sorting: Name

OK Cancel Apply

Output Control

Output Type: Output Opt **自訂輸出內容**

Write Results of All Active mesh sets

Nodal Results

- Displacement Mesh Set...
- Applied Load Mesh Set...
- Reaction Force Mesh Set...
- Grid Point Force Mesh Set...
- Contact Mesh Set...

Element Results

- Force Mesh Set...
- Stress Mesh Set...
- Strain Mesh Set...
- Status Mesh Set...
- Multi-layered Grid Mesh Set...

Output Option

Binary Binary and Text

Element Output Location

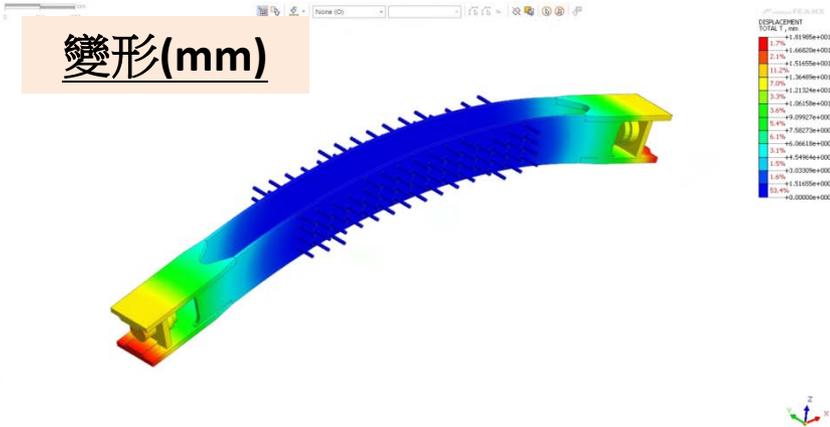
- Element Corner Results
- Shell Mid-Plane Results

Number of Beam Output Segments: 4

OK Cancel

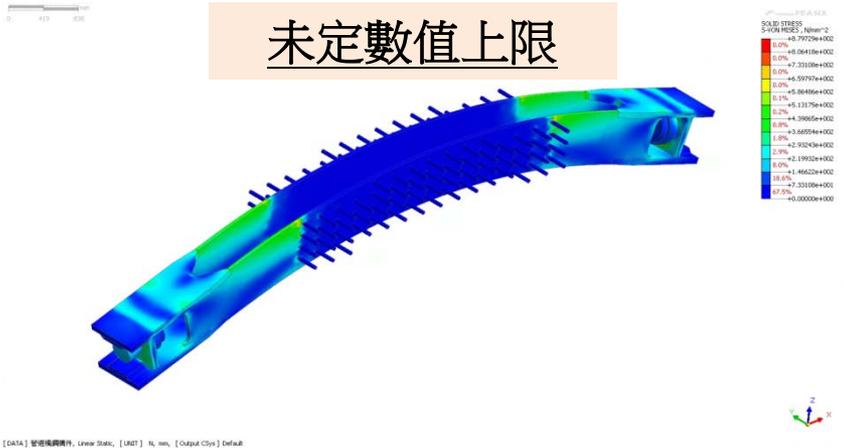
線性分析結果

變形(mm)

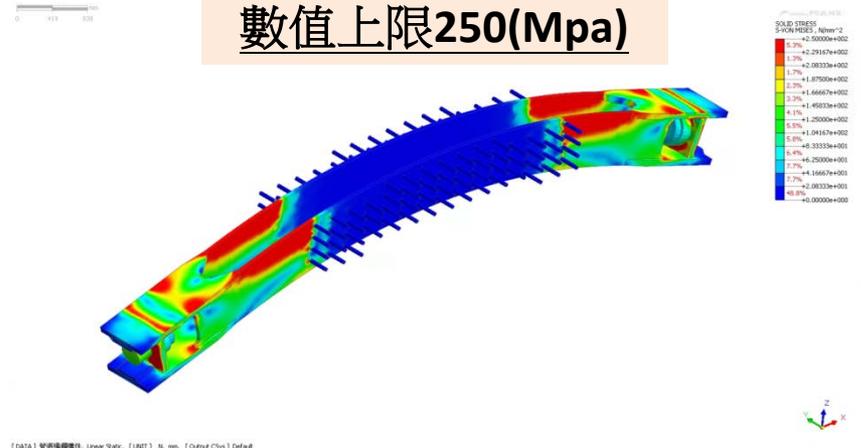


VON MISES Stress (MPa)

未定數值上限



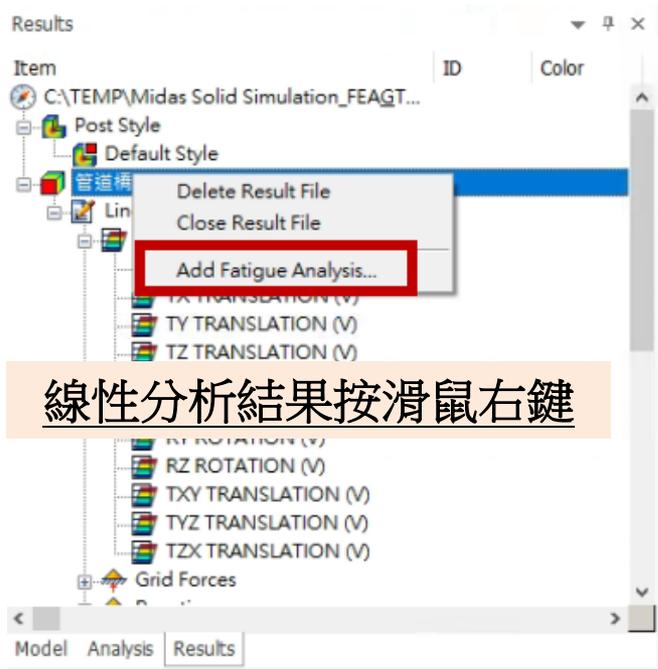
數值上限250(Mpa)



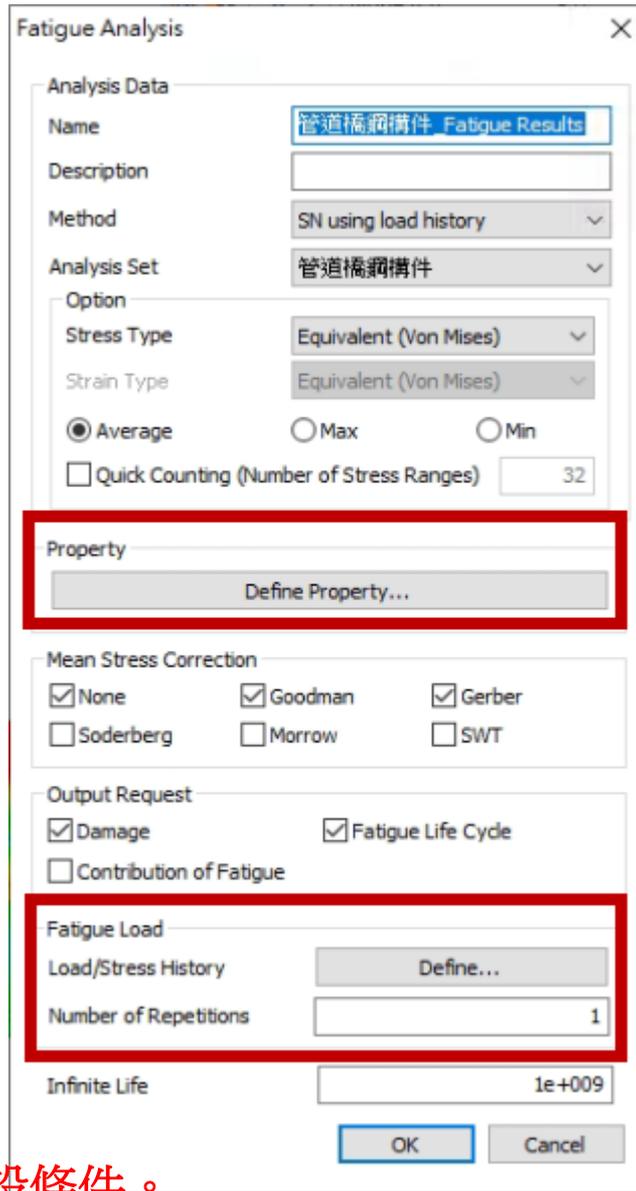
註:範例相關參數使用假設條件。



疲勞分析-1



線性分析結果按滑鼠右鍵



材料疲勞參數

定義加載曲線

註:範例相關參數使用假設條件。

疲勞分析-2

材料疲勞參數

Property

Define Property...

疲勞參數(假設)

Material: 1: A36

Yield Strength: 255 N/mm²

Tensile Strength: 490 N/mm²

Direct Input

Endurance Limit: 207 N/mm²

Cycles at Endurance: 1000000

Function

S-N Curve: [] [...]

Add Modify Delete

No	Material	Type
1	A36	Direct Input

Close

註:範例相關參數使用假設條件。



疲勞分析-3

Fatigue Analysis

Analysis Data

Name: 鏈接鉗箱系統_Fatigue Results

Description:

Method: SN using load history

Analysis Set: 鏈接鉗箱系統

Option

Stress Type: Equivalent (Von Mises)

Strain Type: Equivalent (Von Mises)

Average Max Min

Quick Counting (Number of Stress Ranges) 32

Property

Define Property...

Mean Stress Correction

None Goodman Gerber

Soderberg Morrow SWT

Output Request

Damage Fatigue Life Cycle

Contribution of Fatigue

Fatigue Load

Load/Stress History: Define...

Number of Repetitions: 86400

Infinite Life: 1e+009

1天=86400sec/10sec=8640

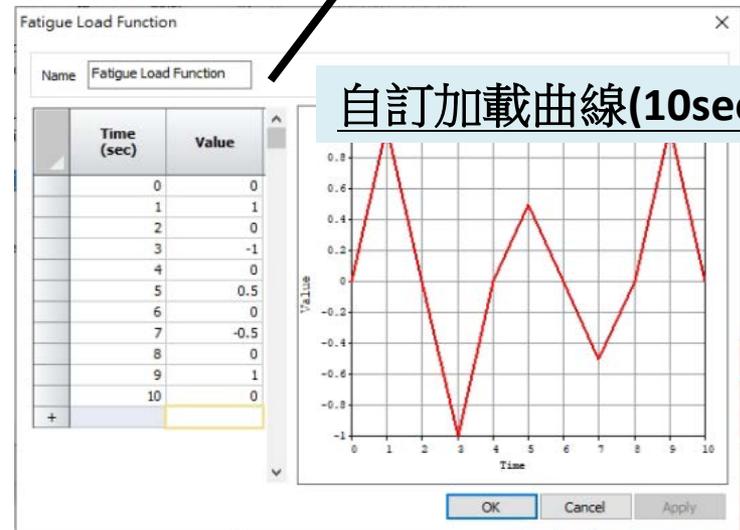
Define Load/Stress History

濃度因子

Name	Concentration Factor	Fatigue Load Function
Linear Static	1.0000	Fatigue Load Function

指定分析工況

Define Fatigue Load Function Reset OK Cancel

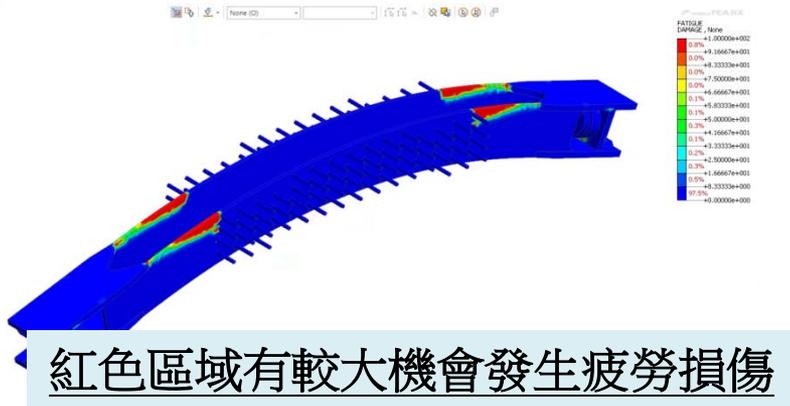


註:範例相關參數使用假設條件。

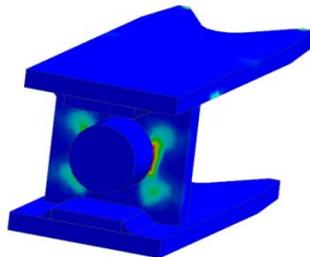
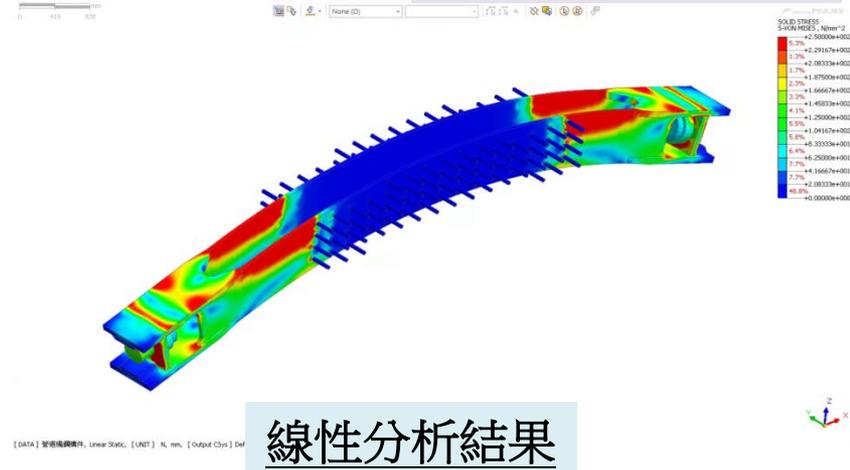


疲勞分析結果

疲勞分析結果



上限定成250(MPa)



註:範例相關參數使用假設條件。

