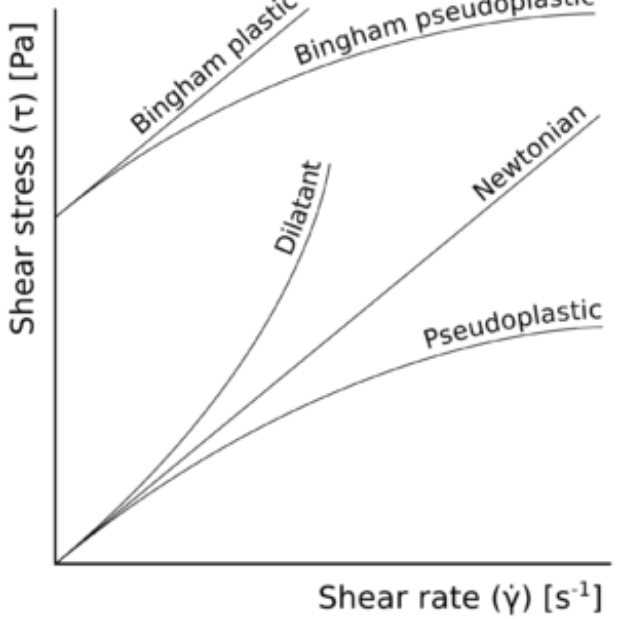
CFD_標準教學系列

強制對流分析1-2D對流分析

台灣邁達斯

流體-1



流體

承受剪應力時將會發生連續變形的物體，包括氣體和液體。

- 液體:可以流動或擴散，但有一定體積。
- 氣體:可以擴散，體積不受限制，沒有固定體積。

流體之種類

- 牛頓流體:應力與應變率成正比，此比例係數為流體的黏度。
- 非牛頓流體
  - 純黏性(非時變性)
    - 剪切增稠流體(脹流性):剪應力增加到某一個數值時，液體中形成了新的結構，引起了阻力的增加，導致液體的表觀黏度增大，同時伴隨著體積的脹大的現象。Ex:陶瓷泥漿，糖果配料，玉米澱粉
    - 剪切稀化流體(假塑性):流體的黏度隨剪應變率的增加而減小。Ex:熔岩、番茄醬、生奶油、血液、顏料、指甲油、巧克力醬
  - 純黏性(時變性):
    - 震凝性流體:剪應變隨著時間增加。
    - 觸變性流體:剪應變隨著時間減少。
  - 黏彈性
    - 賓漢流體:低應力下表現為剛性體，但在高應力下，它會像粘性流體一樣流動，且其流動性為線性的。Ex:牙膏

註:相關內容參考https://zh.wikipedia.org/zh-tw/

流體-2

不可壓縮流 Incompressible Flow

儘管所有氣流都是可壓縮的，當馬赫數Ma<0.3時，流體所受的壓力不足以壓縮流體，僅會造成流體的流動。在此狀況下，流體密度不會隨壓力而改變，此種流場稱為次音速流(Subsonic flow)，流場可視為不可壓縮流場。

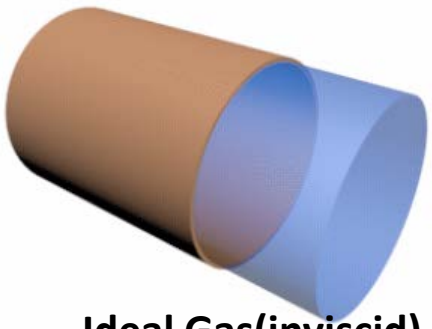
可壓縮流 Compressible Flow

流體在高速運動(流速接近音速或大於音速)時，流體密度會隨壓力而改變，此時氣體之流動稱為可壓縮流場(Compressible flow)。

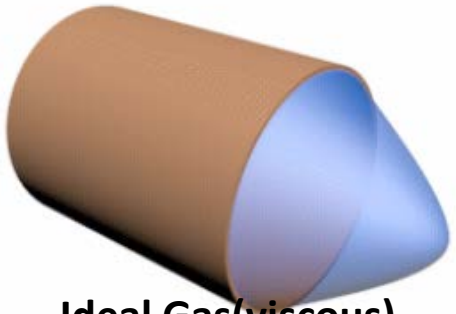
- Pressure Based Compressible:用於不可壓縮和輕度可壓縮的流動。
- Density Based Compressible:基於密度的方法，用於高速可壓縮流。
  - Ideal Gas(inviscid):

    無黏滯性的液體在管中流動，各處流動的速度都相等。
  - Ideal Gas(viscous):

    黏度大於零的液體在管中流動，接近軸心的部分流動的速度最快，靠近管壁速度最慢。



Ideal Gas(inviscid)



Ideal Gas(viscous)

註:圖片內容參考https://highscope.ch.ntu.edu.tw/wordpress/?p=59367

白努利定律

流體力學中，白努利原理(Bernoulli's Principle)指出，不可壓縮流體，在外界不做功的情況下，當液體流速增加時，將會造成液體的壓力或是重力位能減少。
白努利原理其實就是能量守恆定律在流體上的應用。為了滿足能量守恆定律，流體分子力學能的總和應該在流動路徑上的各處皆要相同，亦即動能與位能的和不論流體流往何處皆應保持定值。

$$\frac{1}{2}\rho v^2 + \rho g h + p = \text{constant}$$

v=流體速度
g=重力加速度(地球表面的值約為 9.8 m/s^2)
h=流體處於的深度（從某參考點計）
p=流體所受的壓力強度
P=流體質量密度
constant=常數

註:相關內容參考https://zh.wikipedia.org/zh-tw/

基本觀念

數值分析中，需要空間離散化方法來分析流動或流體傳熱或物質傳輸的控制方程。midas NFX-CFD採用有限元法求解二維和三維問題。

二維元素

可以使用三角形(3個節點)和四面體(4個節點)元素

三維元素

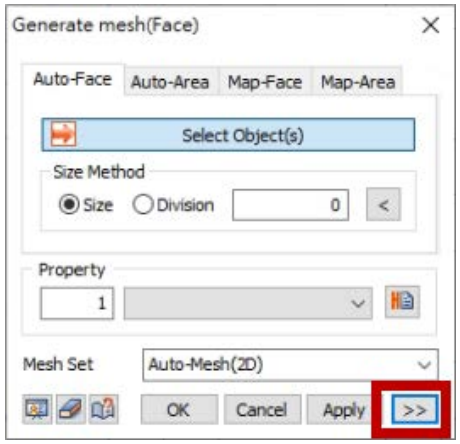
可以考慮具有 4、5、6 或 8 個節點的四面體、五面體或六面體元素
五面體元素可以定義為楔形元素或金字塔形元素。。

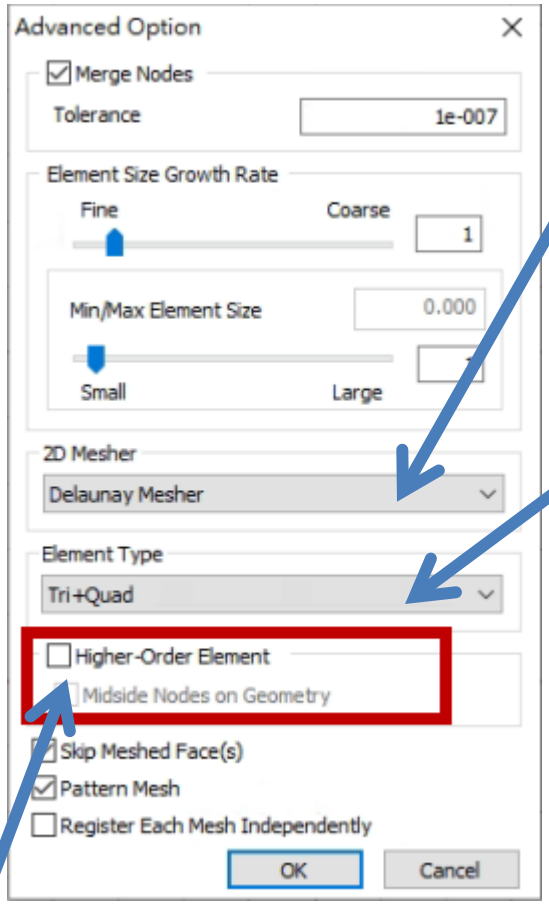
有限增量法(Finite Increment Calculus)

計算對流擴散問題時，有限元素法基於伽遼金方法計算，此方法在計算高雷諾數對流影響，有可能造成數值計算不穩定。NFX-CFD 使用有限增量法 (FIC) 來確保數值的穩定性分析結果。

NFX CFD 2D元素類型

生成網格對話窗右下角點選





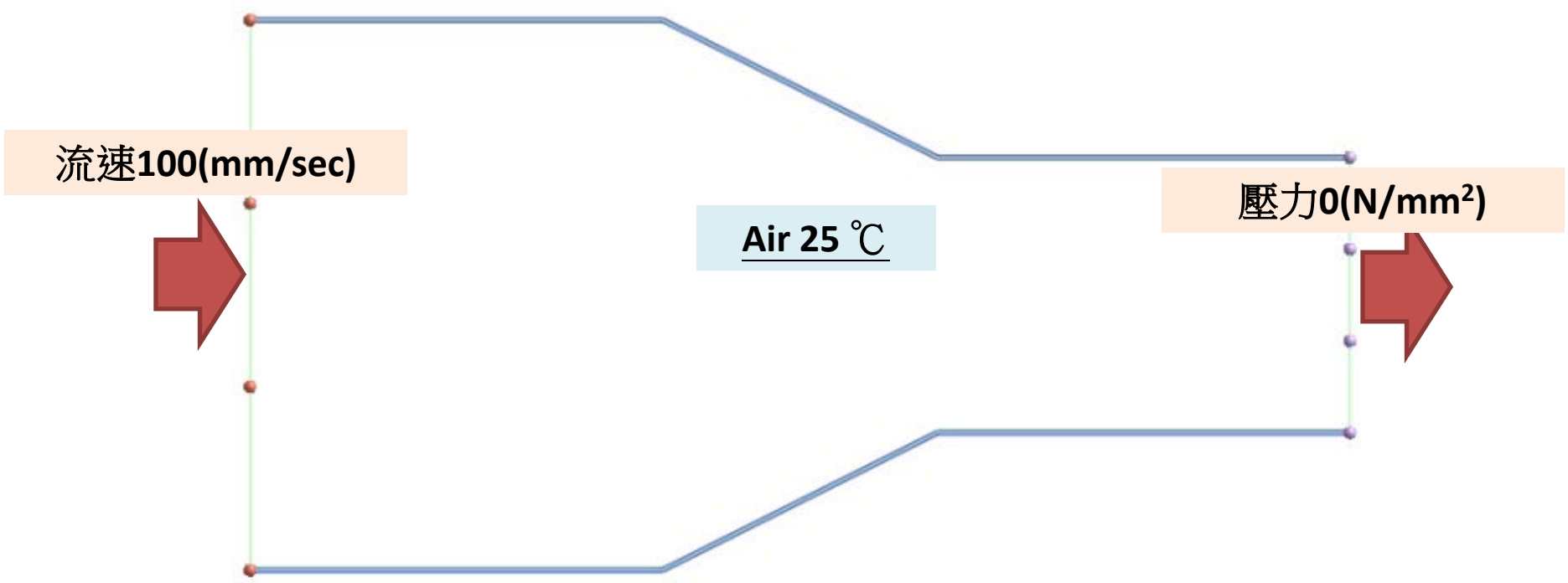
2D元素生成方式
(預設Delaunay網格劃分)

NFX CFD 2D元素類型
(預設三邊形和四邊形混用)

NFX CFD不支援高階元素

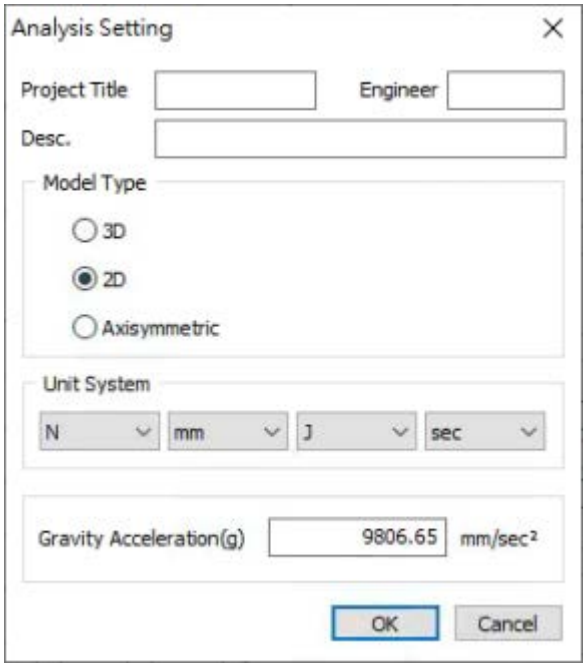
分析說明

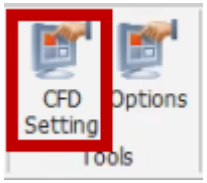
流場分析定義入流和出流相對壓差條件

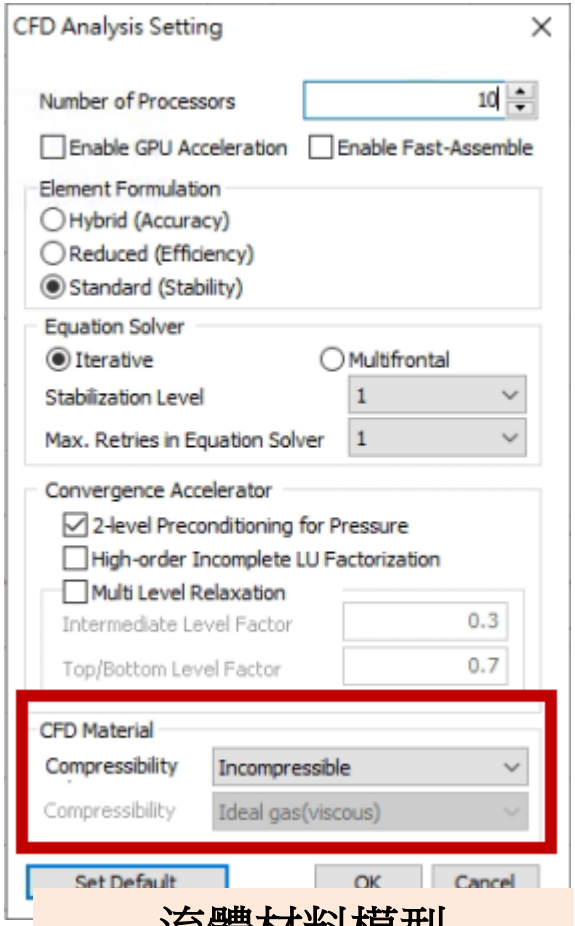


註:NFX CFD預設1大氣壓力，重力方向 (2維空間-y/3維空間-z)

環境

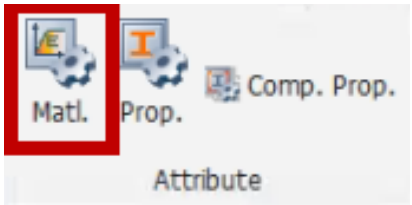


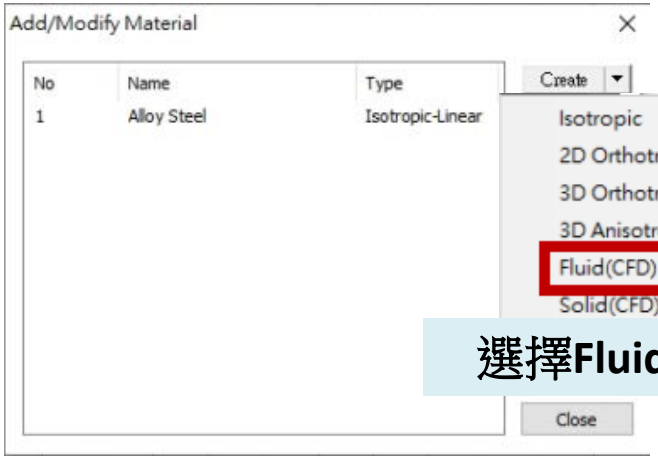




流體材料模型
不可壓縮流

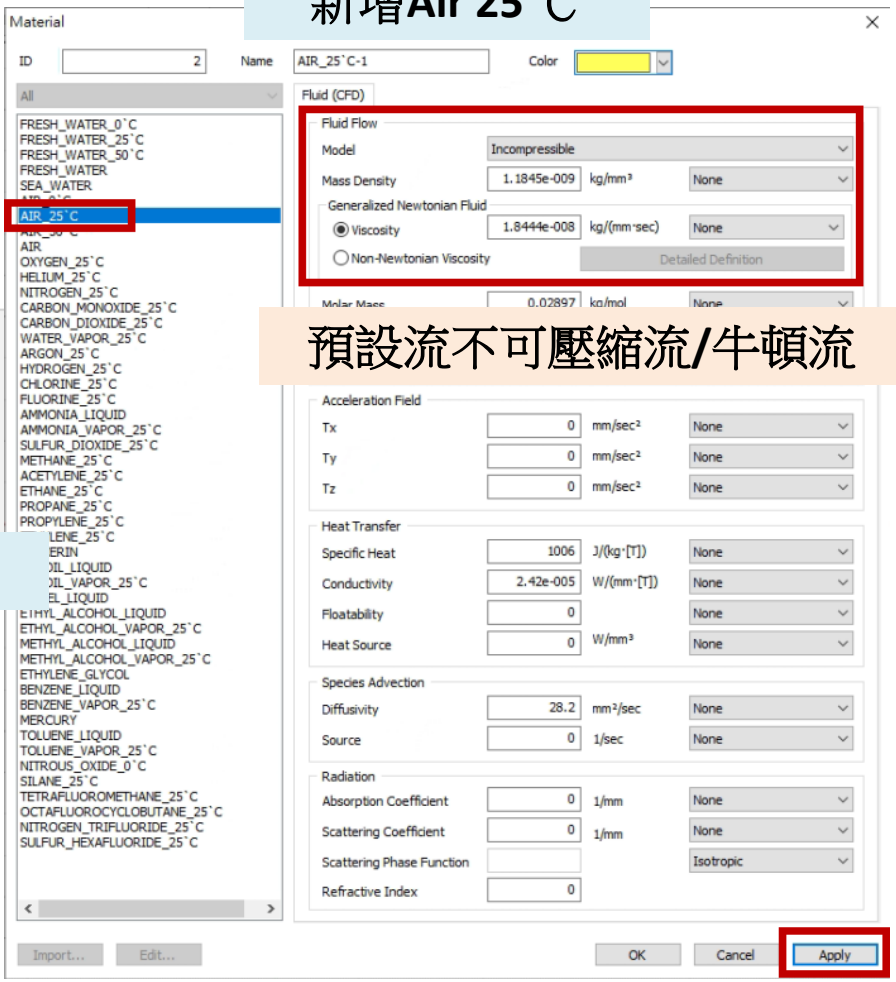
材料





選擇Fluid(CFD)

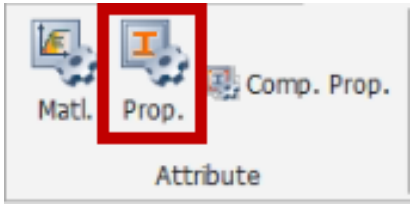
新增Air 25 ℃



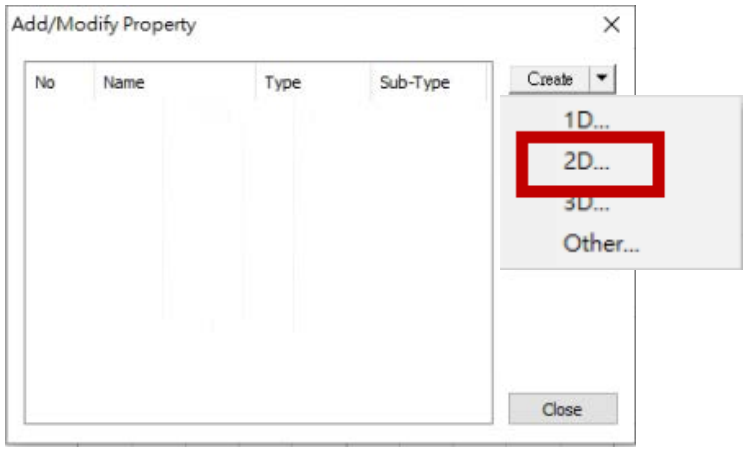
預設流不可壓縮流/牛頓流

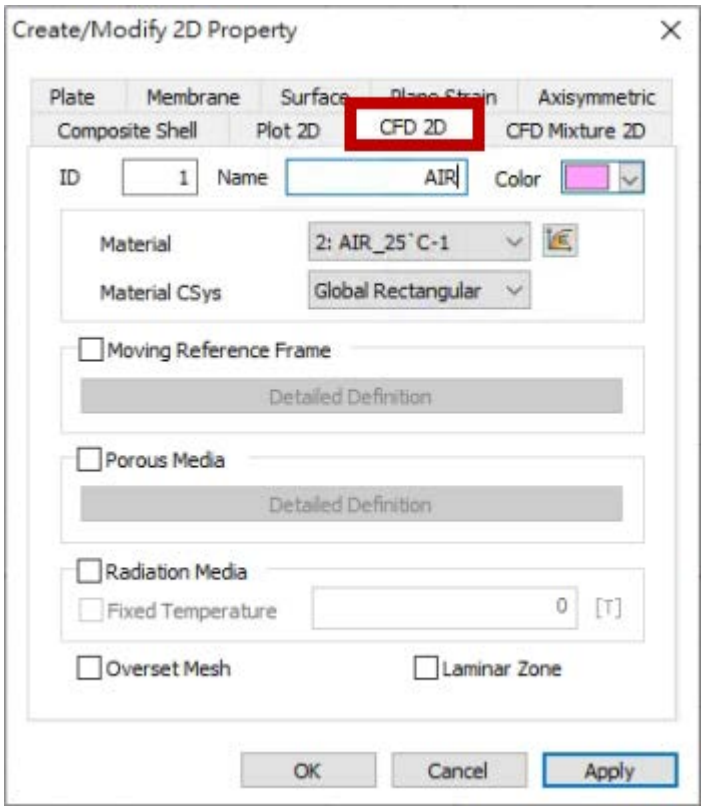
註:NFX CAE/CFD是整合性軟體,材料和屬性包含結構/熱/流場。

屬性



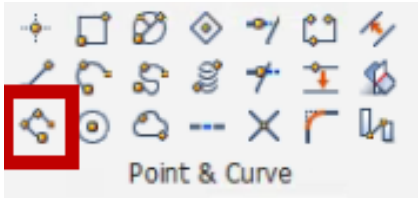
選取CFD 2D



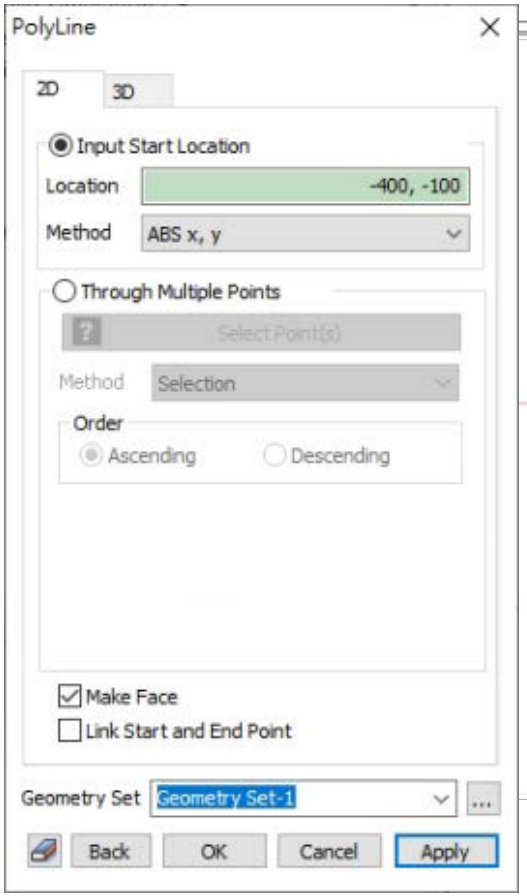


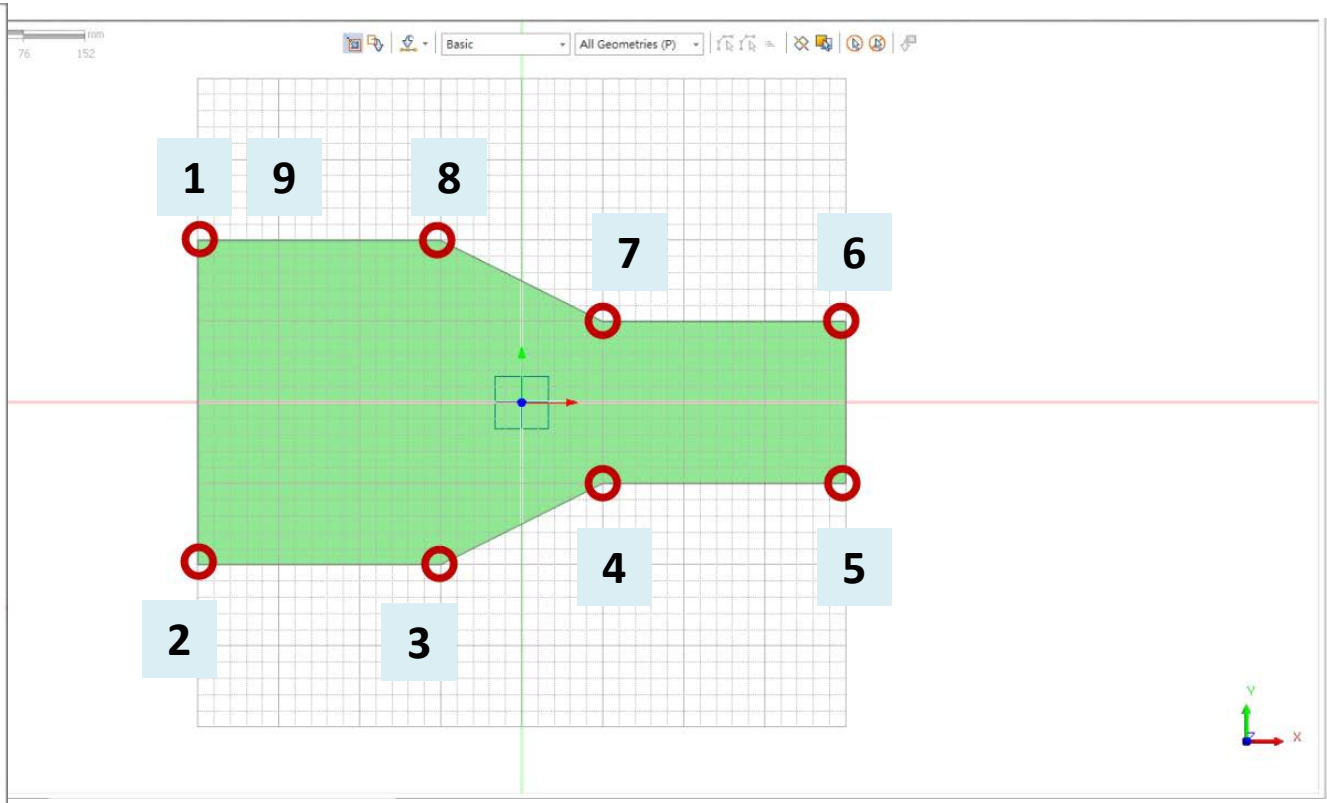
註:NFX CAE/CFD是整合性軟體,材料和屬性包含結構/熱/流場。

幾何繪製

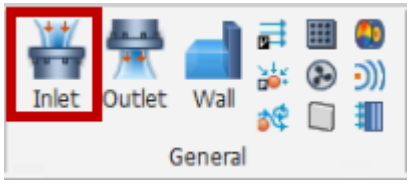

Point & Curve

多折線繪製

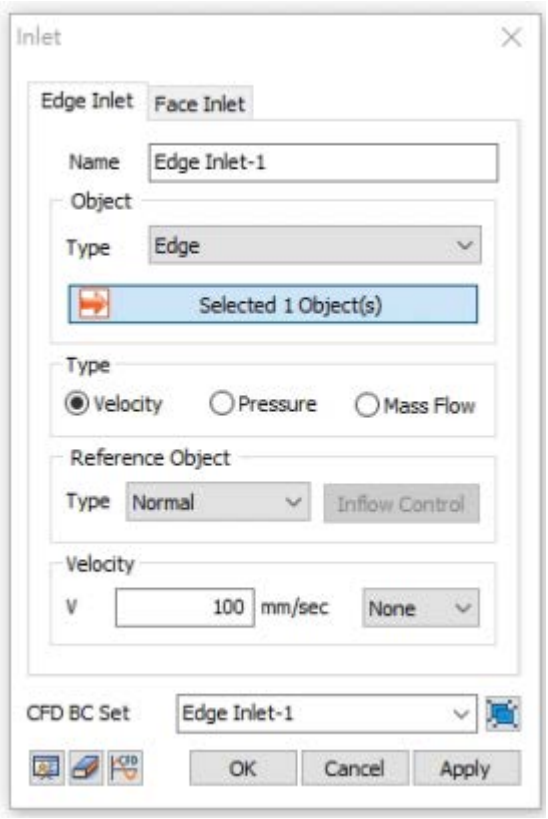

PolyLine
2D
3D
Input Start Location
Location
-400, -100
Method
ABS x, y
Through Multiple Points
Select Point(s)
Method
Selection
Order
Ascending
Descending
Make Face
Link Start and End Point
Geometry Set
Geometry Set-1
Back
OK
Cancel
Apply


Basic
All Geometries (P)
1
9
8
7
6
4
5
2
3

入流



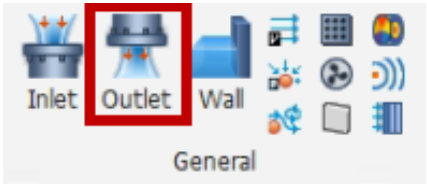
Inlet條件垂直施加在特徵面

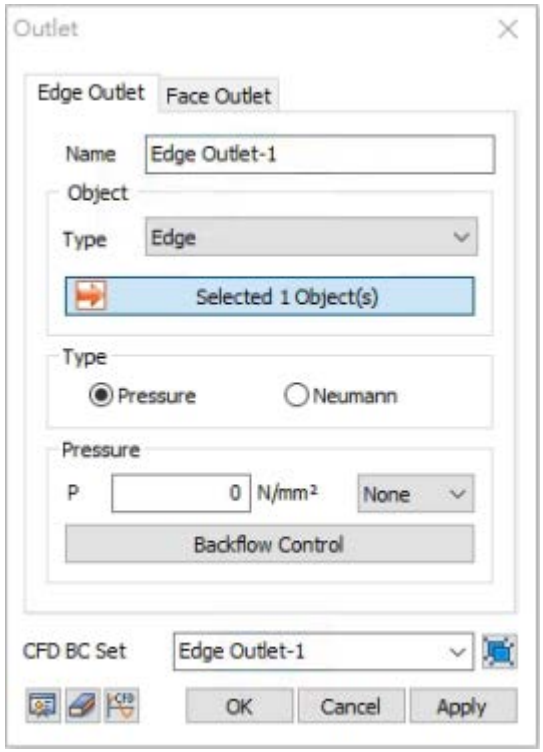


流速100(mm/sec)

註:不同邊界&載荷建議使用不同集合名稱。

出流

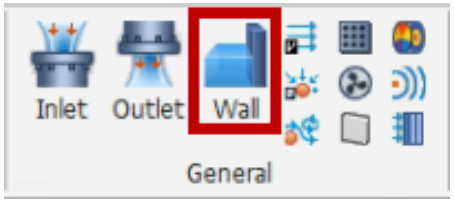




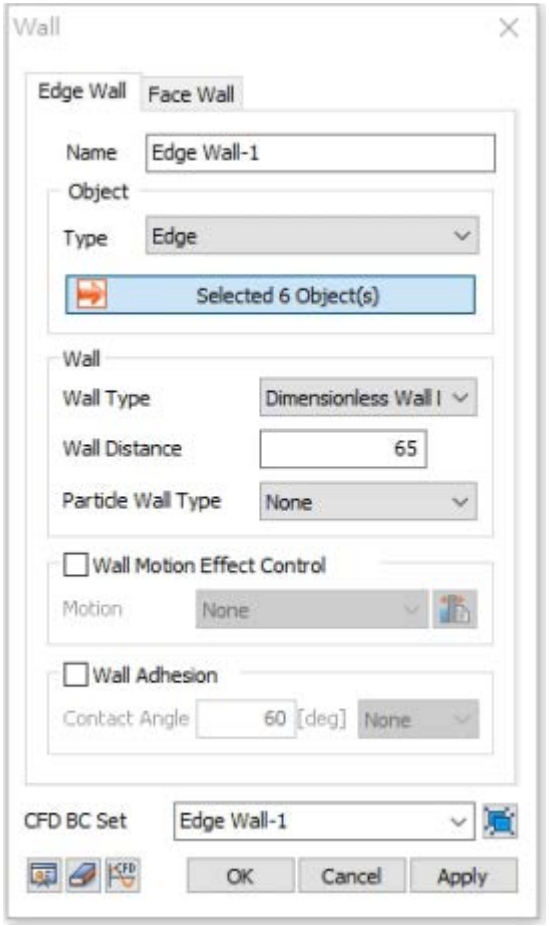
壓力0(N/mm²)

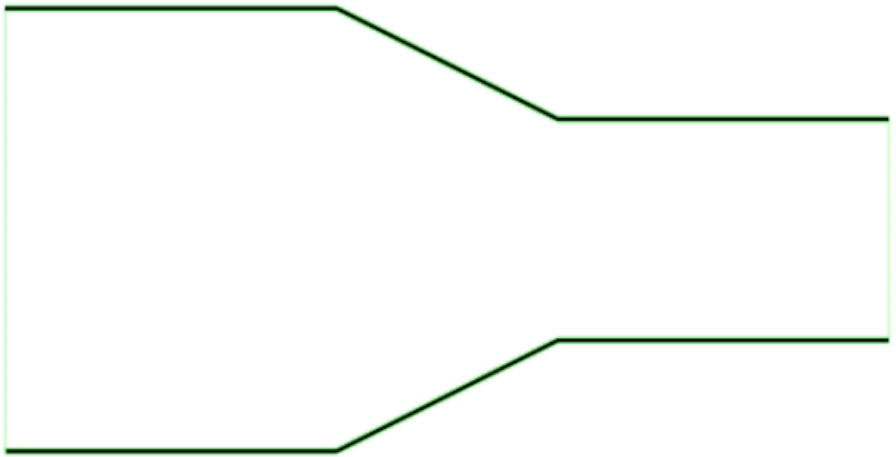
註:不同邊界&載荷建議使用不同集合名稱。

壁



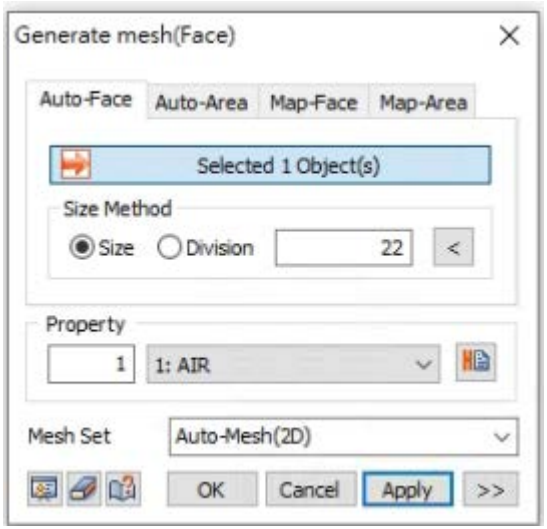
Wall Function 定義流體和壁面關係

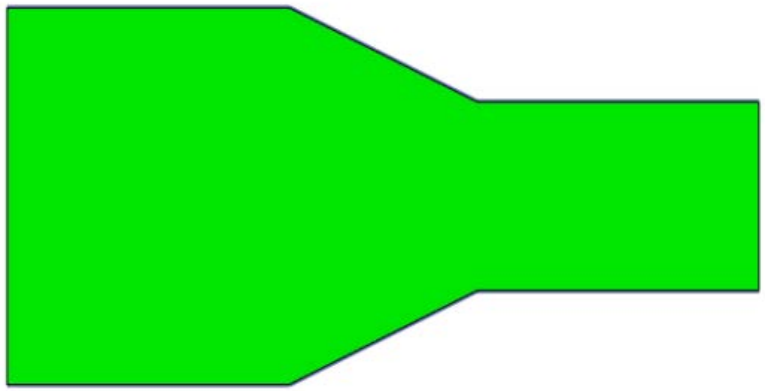


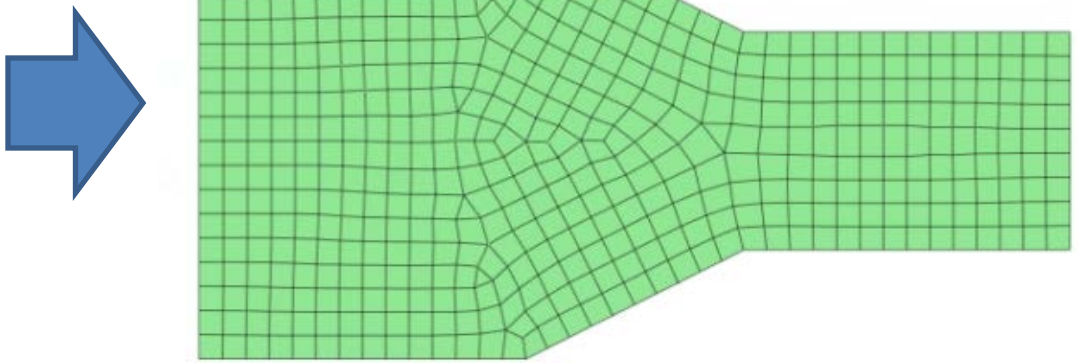


2D網格

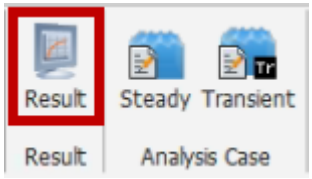

1D
2D
3D
2D->3D
Remesh
Generate


Generate mesh(Face)
Auto-Face
Auto-Area
Map-Face
Map-Area
Selected 1 Object(s)
Size Method
Size
Division
22
<
Property
1
1: AIR
Mesh Set
Auto-Mesh(2D)
OK
Cancel
Apply
>>

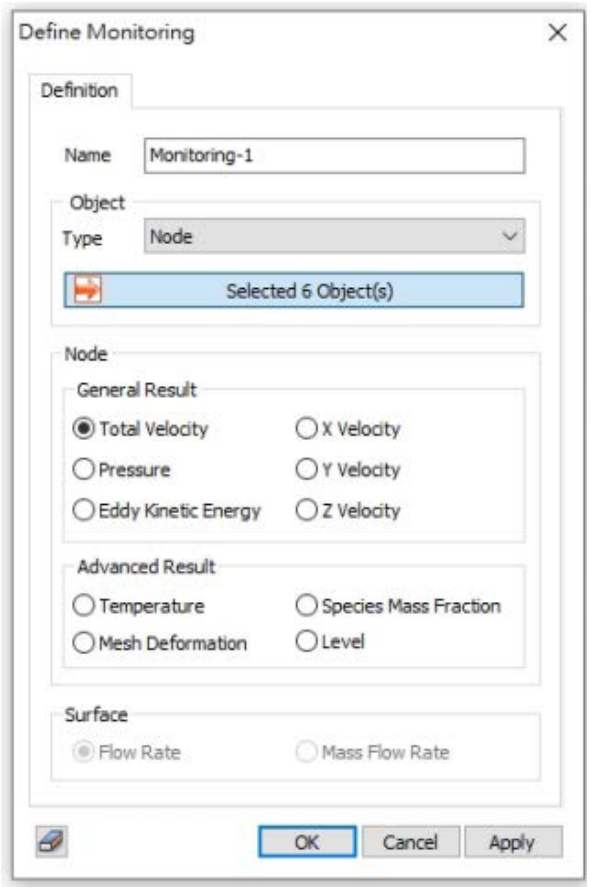




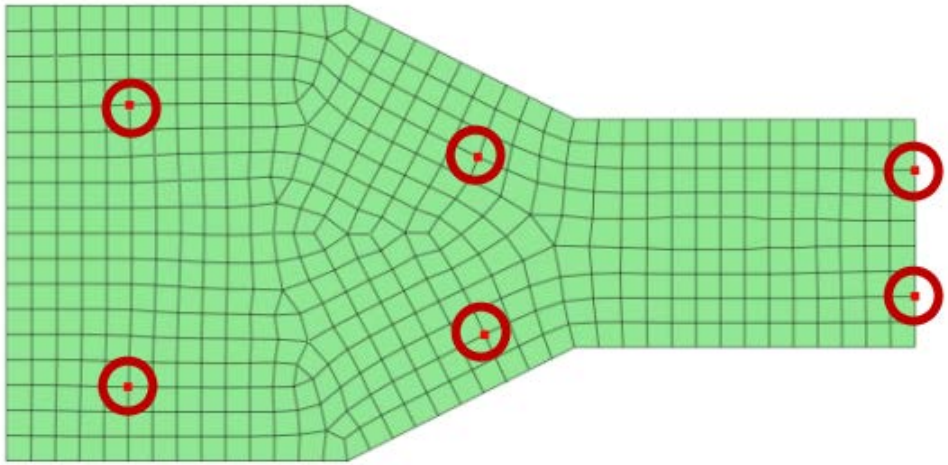
觀測點



輸出特定節點數據同時能協助判斷收斂穩定

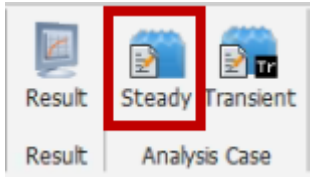


選取流場節點

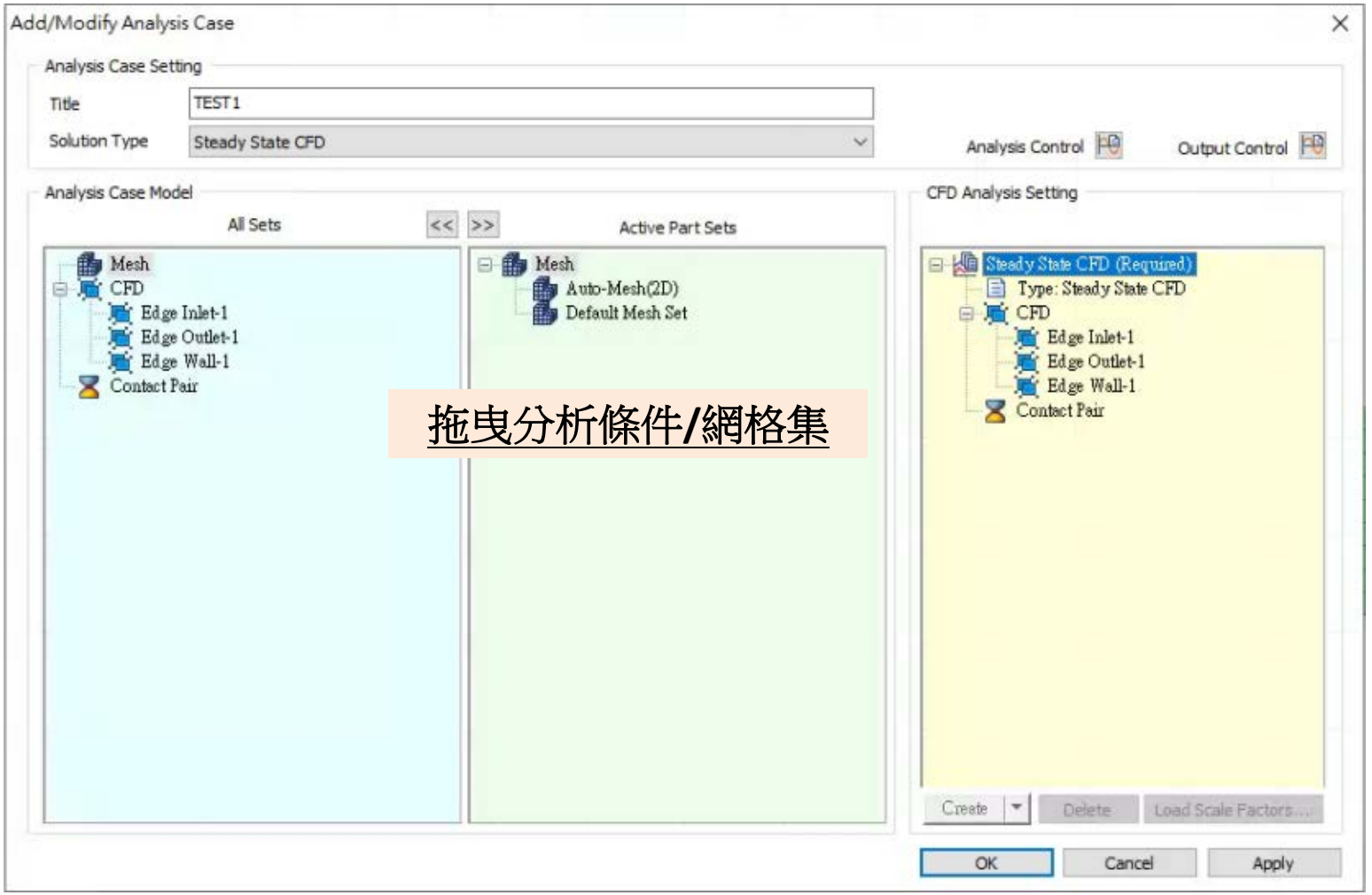


midas NFX-CFD

CFD穩態分析-1

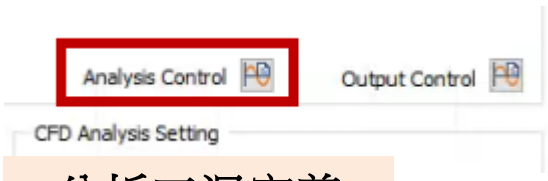


Solution Type : Steady State CFD



拖曳分析條件/網格集

CFD穩態分析-2



分析工況定義

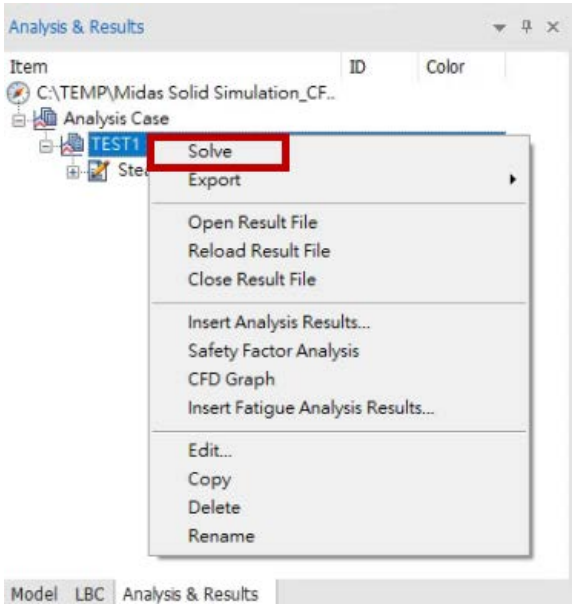
CFD分析模組

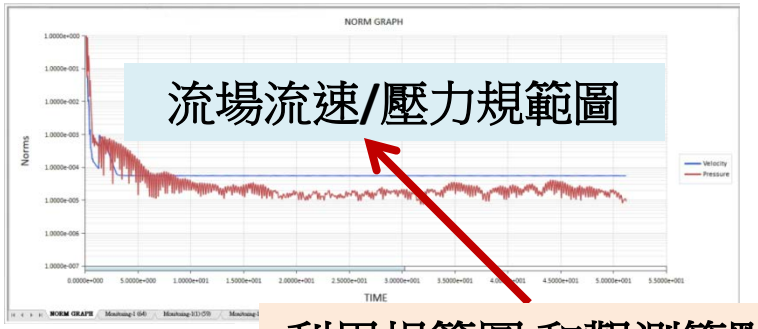
增量步和收斂定義
(未自訂收斂標準預設所有增量全部計算)

建議收斂規範值
(低於此值不會強制終止計算)

大氣壓力/重力方向

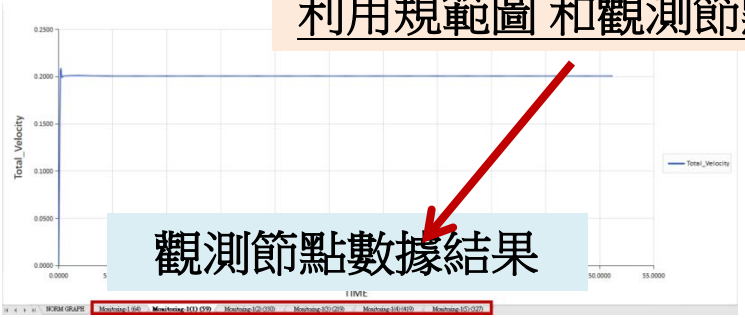
執行計算



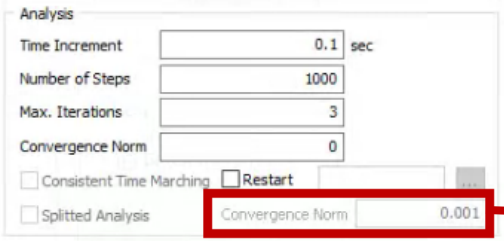


流場流速/壓力規範圖

利用規範圖 和觀測節點判斷收斂



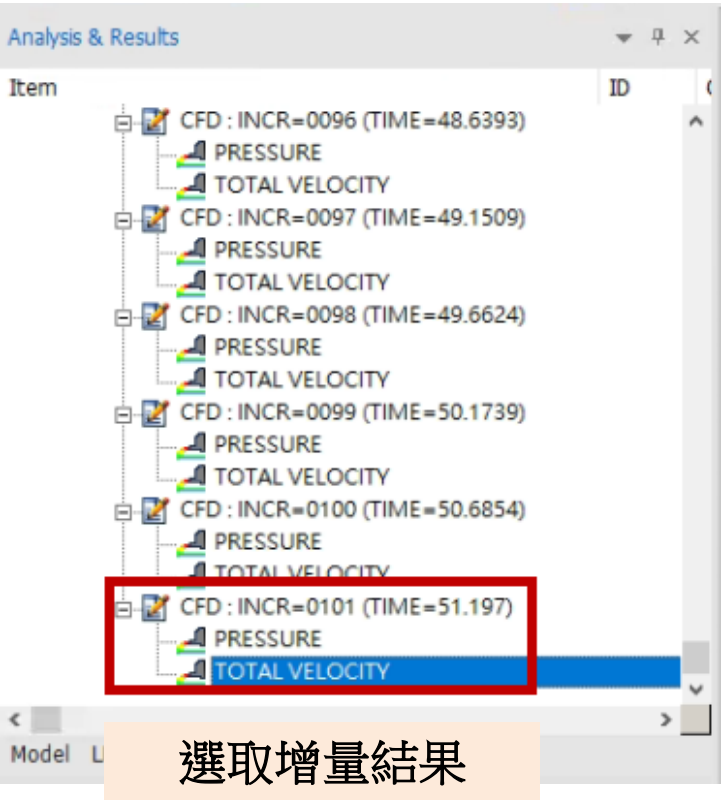
觀測節點數據結果



利用輸出文件判斷收斂

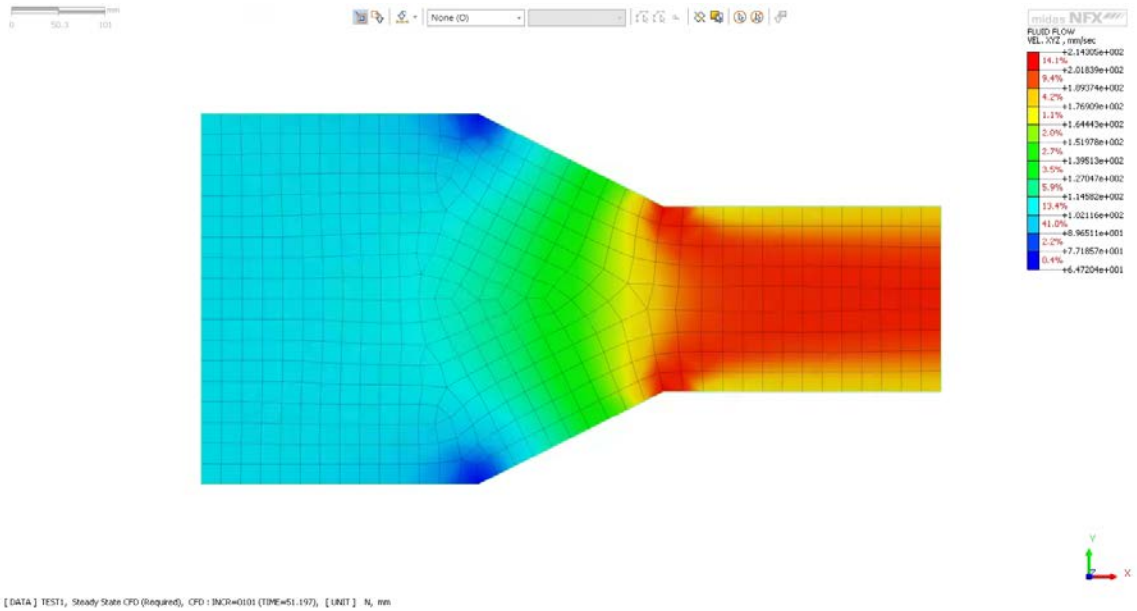
建議收斂規範值
(低於此值不會強制終止計算)

分析結果

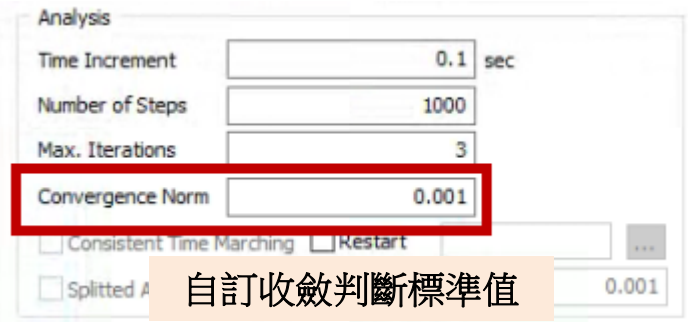


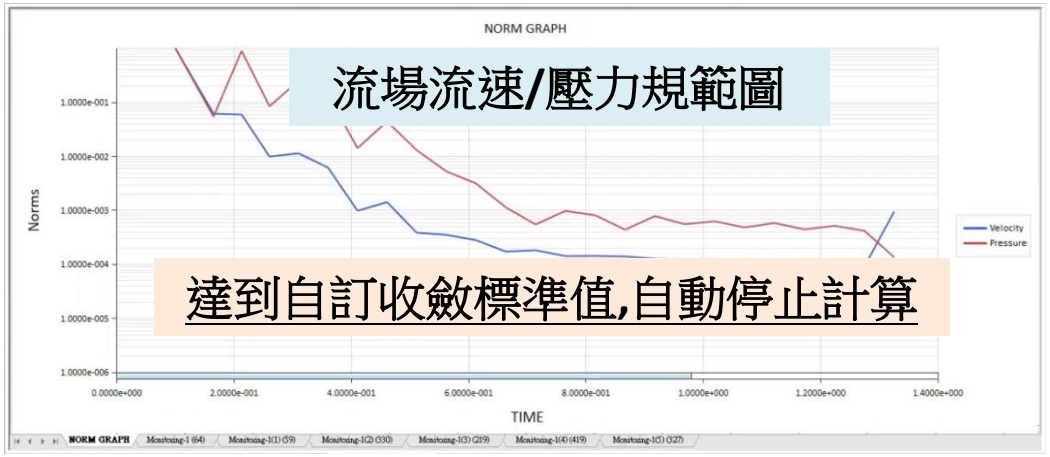
選取増量結果

流速(mm/sec)



自動收斂判斷


Analysis
Time Increment
0.1
sec
Number of Steps
1000
Max. Iterations
3
Convergence Norm
0.001
Consistent Time Marching
Restart
Splitted A
0.001
自訂收斂判斷標準值


NORM GRAPH
流場流速/壓力規範圖
Norms
TIME
Velocity
Pressure
達到自訂收斂標準值,自動停止計算


Output
> WARNING [4007] : MORE THAN 3 ITERATIONS MAY BE NEEDED.
> STEP = 12, FLOW TIME = 6.632E-01 (DT = 5.07E-02), ITER = 3, NORM = VEL(1.731E-04) PRES(1.145E-03)
> WARNING [4007] : MORE THAN 3 ITERATIONS MAY BE NEEDED.
> STEP = 13, FLOW TIME = 7.139E-01 (DT = 5.07E-02), ITER = 2, NORM = VEL(1.832E-04) PRES(5.498E-04) : CONVERGED
> STEP = 14, FLOW TIME = 7.647E-01 (DT = 5.08E-02), ITER = 2, NORM = VEL(1.433E-04) PRES(9.798E-04) : CONVERGED
> STEP = 15, FLOW TIME = 8.156E- 04) PRES(8.149E-04) : CONVERGED
> STEP = 16, FLOW TIME = 8.665E- 04) PRES(4.401E-04) : CONVERGED
> STEP = 17, FLOW TIME = 9.175E- 04) PRES(7.857E-04) : CONVERGED
> STEP = 18, FLOW TIME = 9.684E- 04) PRES(5.557E-04) : CONVERGED
> STEP = 19, FLOW TIME = 1.019E+00 (DT = 5.10E-02), ITER = 3, NORM = VEL(1.177E-04) PRES(6.302E-04) : CONVERGED
> STEP = 20, FLOW TIME = 1.070E+00 (DT = 5.10E-02), ITER = 2, NORM = VEL(1.094E-04) PRES(4.846E-04) : CONVERGED
> WRITING OUTPUT FILE
> STEP = 21, FLOW TIME = 1.121E+00 (DT = 5.10E-02), ITER = 3, NORM = VEL(1.069E-04) PRES(5.854E-04) : CONVERGED
> STEP = 22, FLOW TIME = 1.172E+00 (DT = 5.10E-02), ITER = 2, NORM = VEL(9.957E-05) PRES(4.458E-04) : CONVERGED
> STEP = 23, FLOW TIME = 1.223E+00 (DT = 5.10E-02), ITER = 3, NORM = VEL(9.812E-05) PRES(5.183E-04) : CONVERGED
> STEP = 24, FLOW TIME = 1.274E+00 (DT = 5.10E-02), ITER = 2, NORM = VEL(9.058E-05) PRES(4.219E-04) : CONVERGED
> STEP = 25, FLOW TIME = 1.325E+00 (DT = 5.10E-02), ITER = 1, NORM = VEL(9.539E-04) PRES(1.359E-04) : CONVERGED
> SOLVER TERMINATED : STEADY STATE FOUND IN FIRST ITERATION.
> WRITING OUTPUT FILE
>
> [SYSTEM INFO]
> PROCESSOR TYPE : CPU
> NUMBER OF THREADS : 10
> MAXIMUM MEMORY USAGE : 14 MB
> AVAILABLE MEMORY : 104673 MB
> TOTAL CPU TIME : 24.171 sec
> WALL CLOCK TIME : 2.773 sec
> SOLVER FINISHED AT : 2022/12/01 08:51:43
> TOTAL WARNINGS : 11
>
> Solver was terminated successfully!
輸出文件

CFD_標準教學系列

強制對流分析9-2D邊界層分離

台灣邁達斯

註:範例所需計算時間較長

midas NFX-CFD

邊界層分離
(Separation of Boundary Layers)

當邊界層分離時，其位移厚度會迅速增加，影響外面的位流及壓強場。

案例1.機翼 -外流場

壓強場的變化會讓壓差損失提高，若壓差損失增加太多，飛機出現升力喪失及失速等的情形。

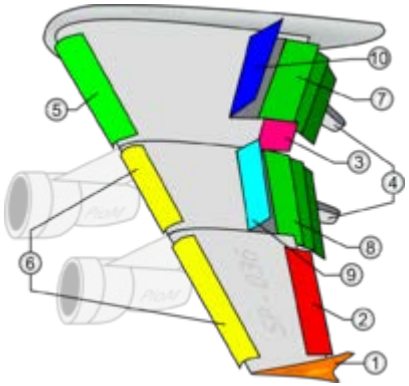
案例2.壓縮機/風機-內流場

邊界層分離會增加流動損失，造成壓縮機/風機喘振(surge)。

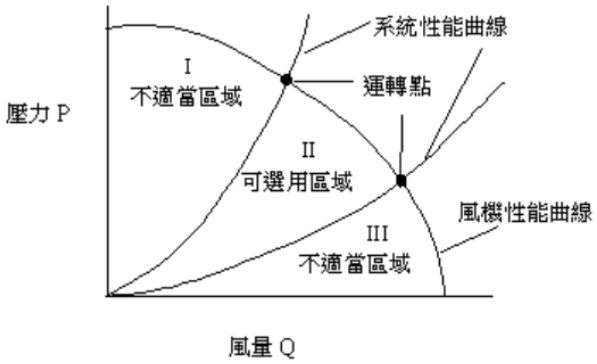
案例3. 橋體-外流場

邊界層分離的另一個影響是渦旋脫落，也就是卡門渦街。當渦旋開始脫離邊界的表面，渦旋會以固定的頻率離開邊界表面。渦旋的周期性脫落會使得附近的結構產生振動，當振動頻率接近共振頻率，會造成嚴重的結構破壞。

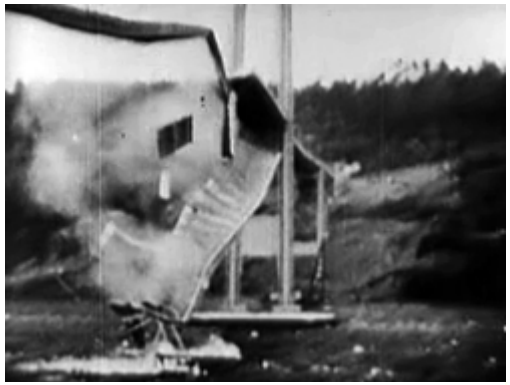
機翼與附屬裝置



風機性能曲線



塔科馬海峽吊橋崩塌



Reference

https://zh.wikipedia.org/wiki/邊界層分離

卡門渦街

(Karman Vortex Street)

流體中安置的阻流體，在特定條件下會出現不穩定的邊界層分離，阻流體下游的兩側，會產生兩道非對稱地排列的旋渦，其中一側的旋渦循時針方向轉動，另一旋渦則反方向旋轉，這兩排旋渦相互交錯排列，各個旋渦和對面兩個旋渦的中間點對齊，如街道道兩邊的街燈一般，故名渦街。

卡門渦街形成的條件

卡門渦街的形成條件：對於在流體中的圓柱體雷諾數（$47<Re<10^5$）

(1)雷諾數=30

圓柱體後的液體呈平陸狀態。

(2)雷諾數=40

圓柱體後的液體開始出現正弦式波動。

(3)雷諾數=47

圓柱體後的液體，前端仍然呈正弦狀，後端則逐漸脫離正弦波動。

(4)雷諾數>47

圓柱體後的液體，出現卡門渦街。

(5)雷諾數在50~85

圓柱體後的液體壓力，呈等振幅波動。

(6)雷諾數>185

圓柱體後的液體壓力，呈非均勻振幅波動。

Reference

https://zh.wikipedia.org/wiki/卡門渦街

分析說明

層流Laminar Flow Re:150

Turbulence Model : Laminar

Re:150 ρ:1 (kg/m^3) U:0.2 (m/s)

D:1(m)外部流動取物體主要尺寸

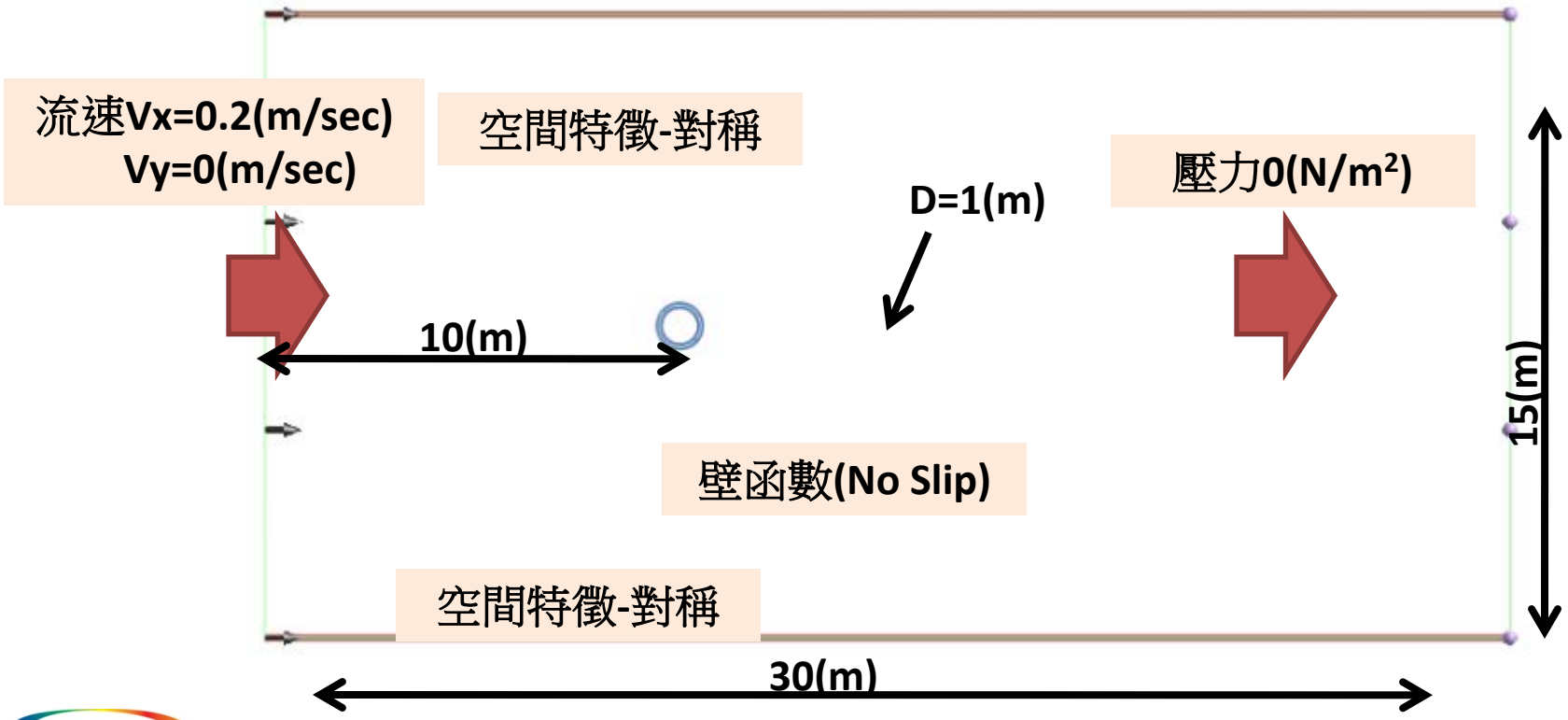
μ: ρ × U ×D/Re=0.00133 (kg/m-s)

δ :0.2(m)邊界層厚度(0.99 × 0.1)

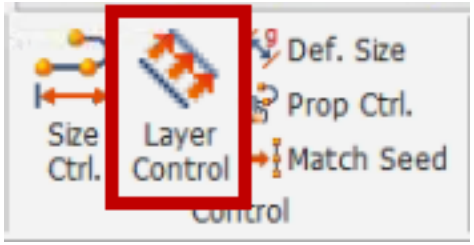
湍流動能(TKE)&湍流尺度

K= 0.000439

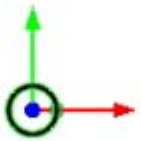
I=0.07



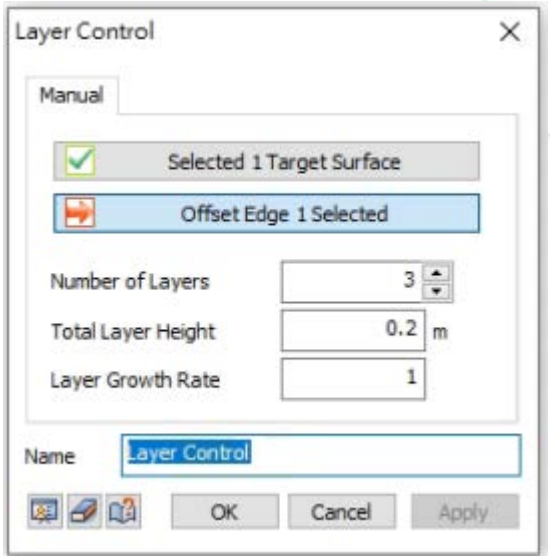
邊界層網格尺寸指定



Step1.選取面特徵



Step2.選取球體線特徵



層流**Laminar Flow**

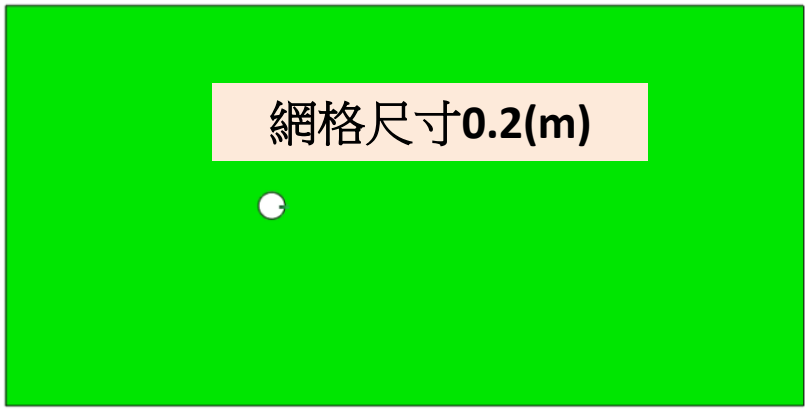
Number of Layers:3

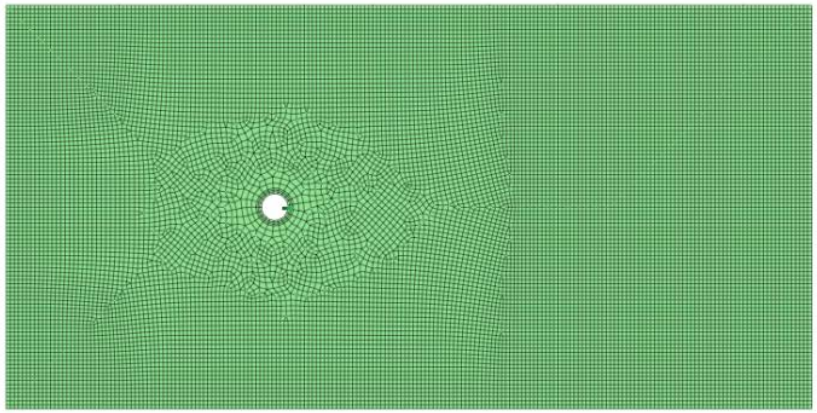
Total Layer Height:0.2(m)

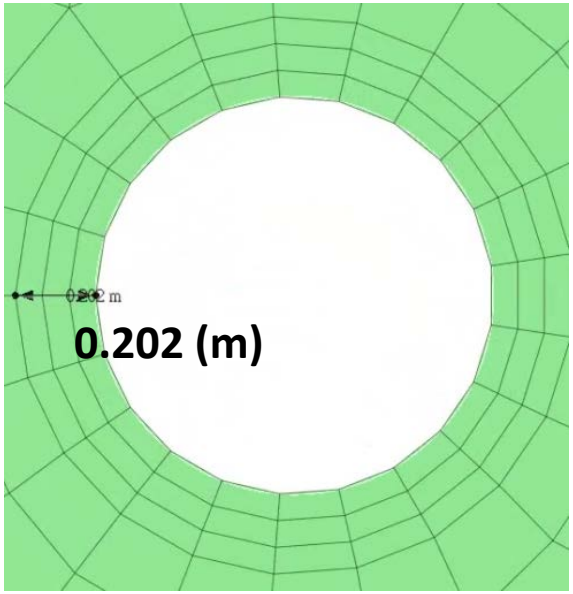
Layer Growth Rate:1

midas NFX-CFD

2D網格







層流**Laminar Flow**

Number of Layers:3

Total Layer Height:0.2(m)

Layer Growth Rate:1

分析說明
(重啓動)

Solution Type : Steday CFD

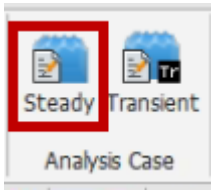
自動寫出重啓動文件

Solution Type : Transient CFD

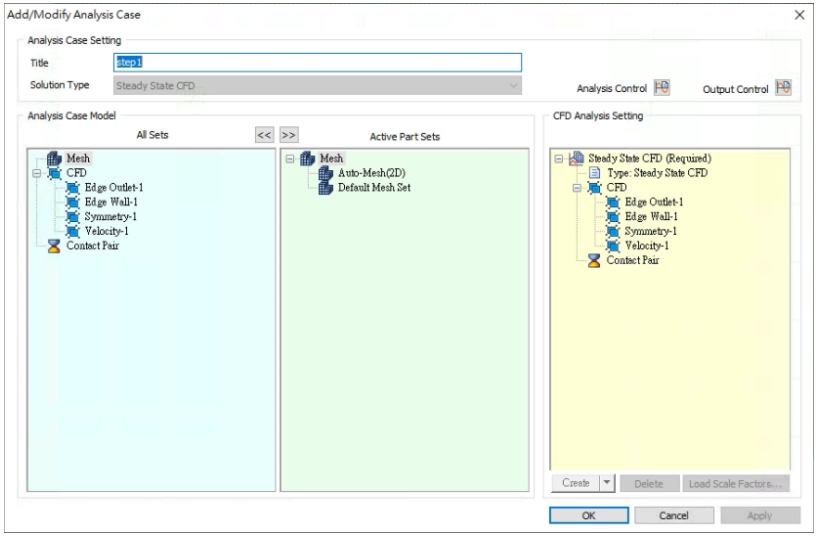
匯入重啓動文件

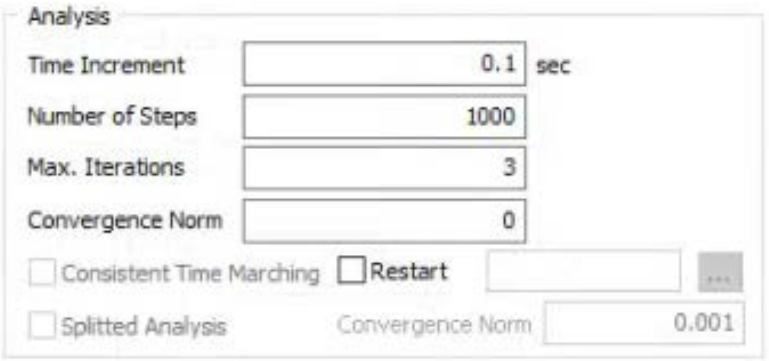
***.RST是NFX重件動文件**

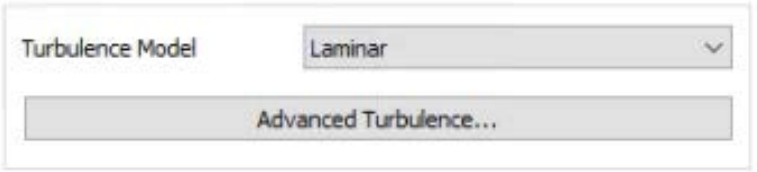
Step1 CFD穩態分析

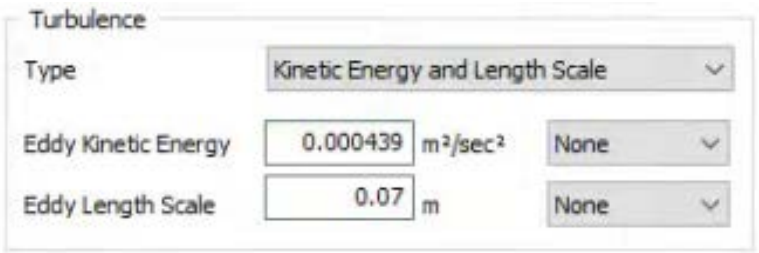


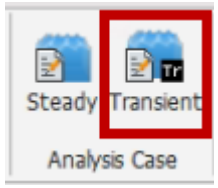
Solution Type : Steady State CFD





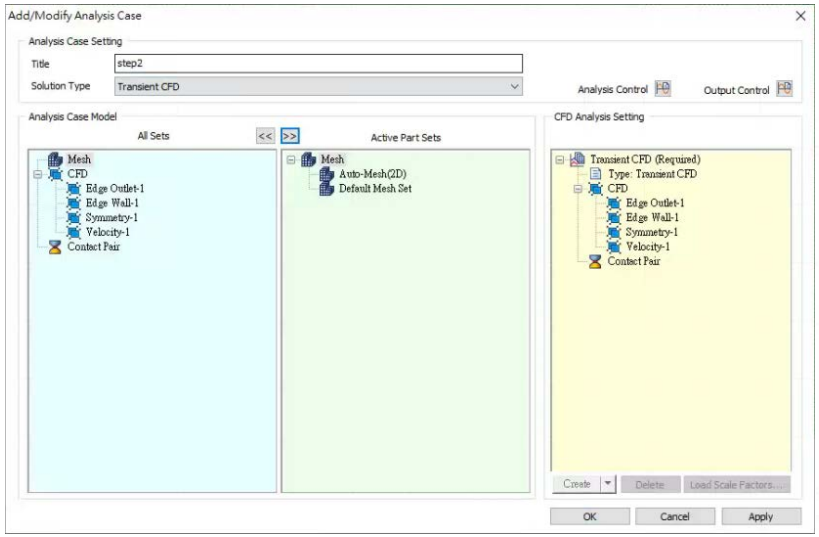


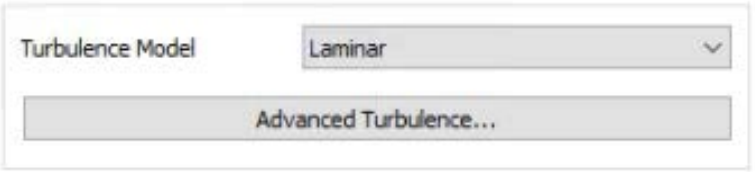


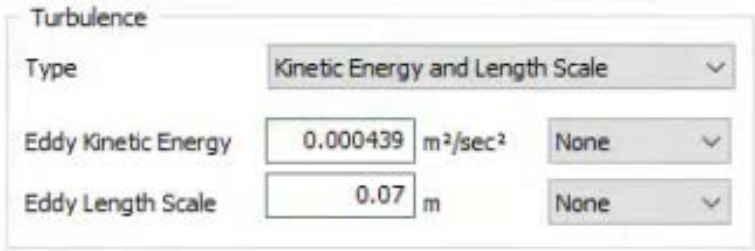


Step2 CFD瞬態分析

Solution Type : Transient CFD







增量步和收斂定義

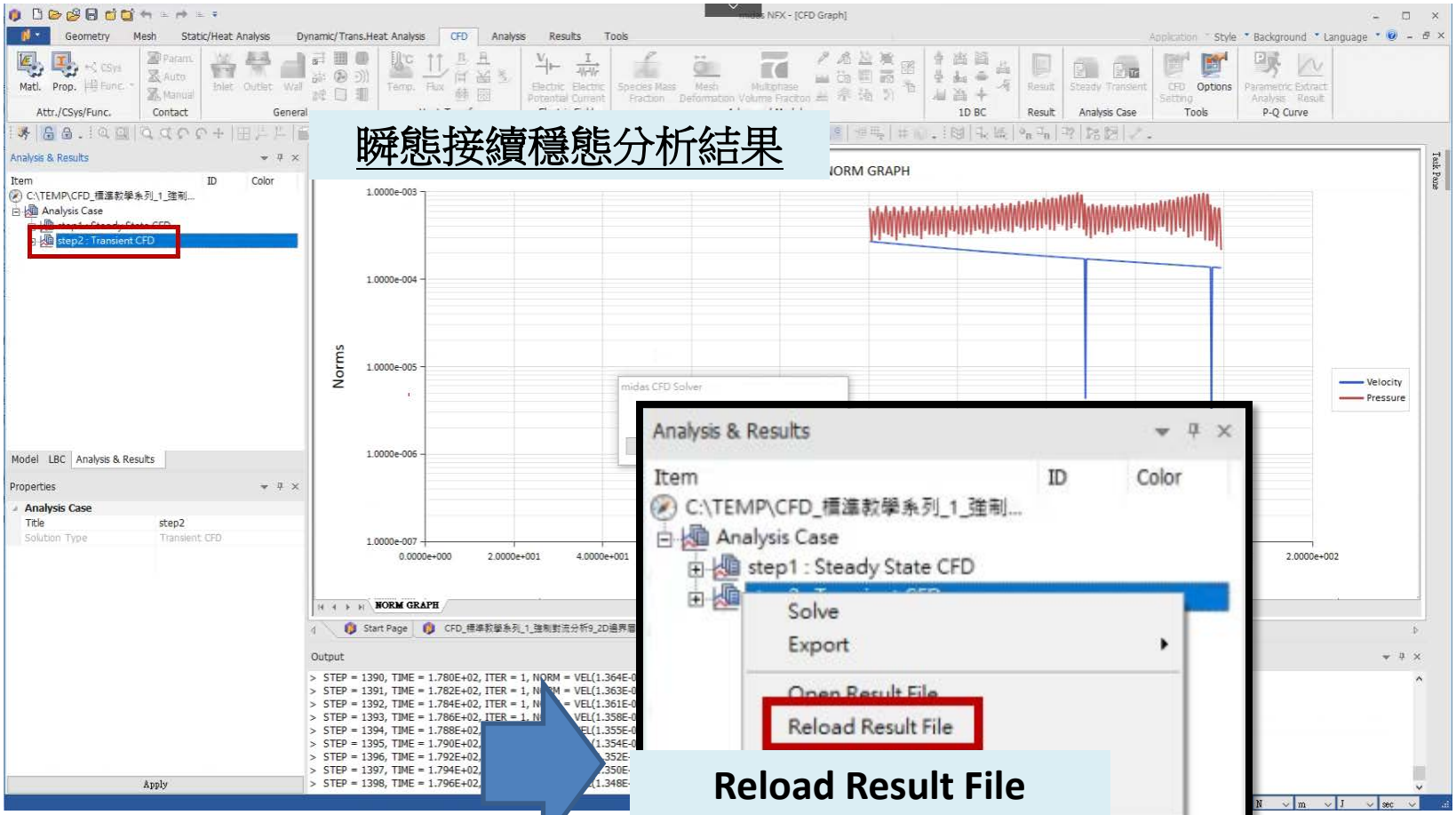
Time Increment :0.2

Number of steps:5000

Interval:50

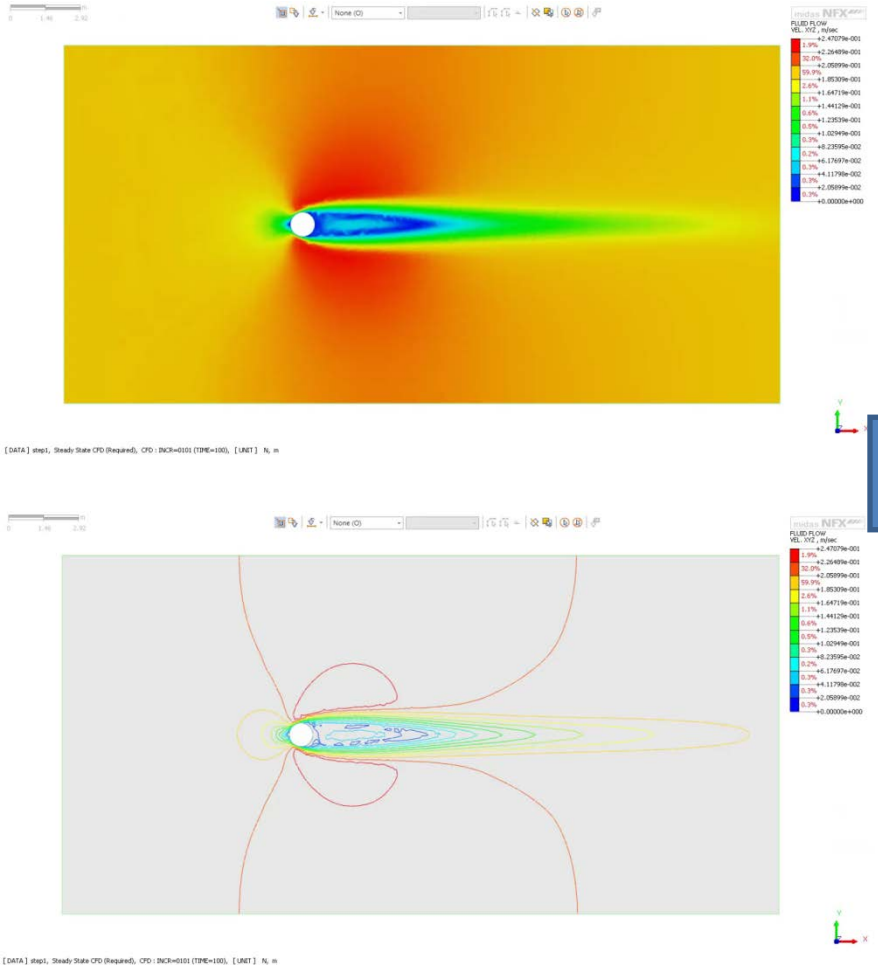
選擇重啓動文件檔Step1.rst

執行計算
(計算時間長)

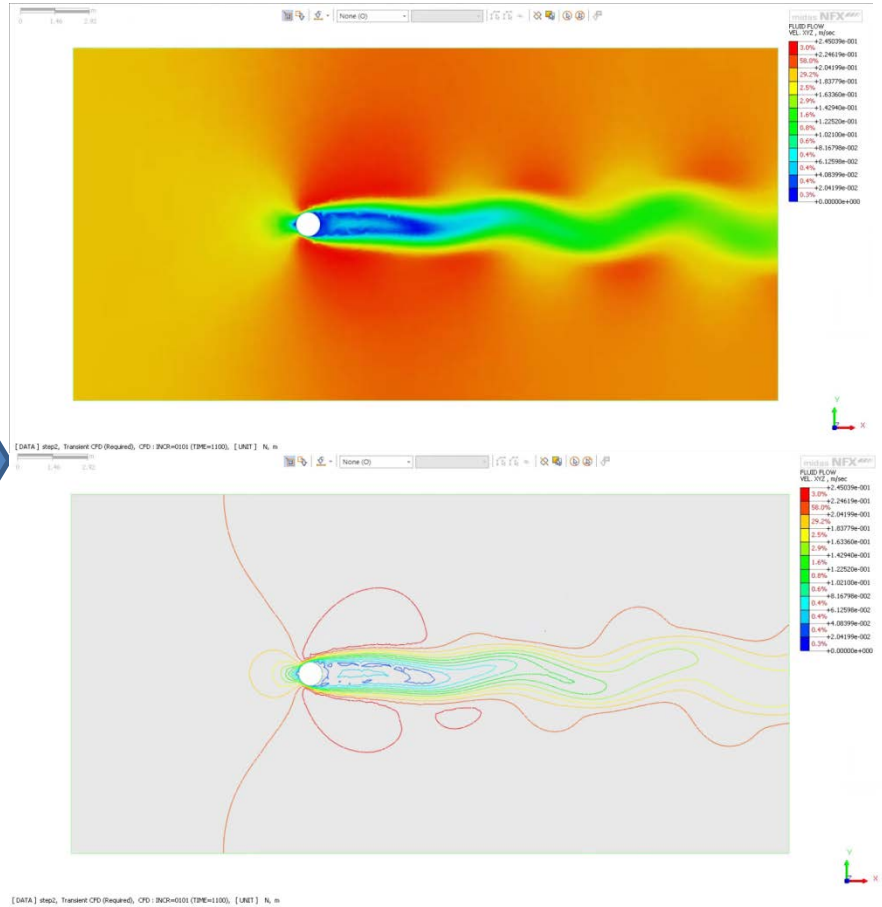

瞬態接續穩態分析結果
Analysis & Results
Item
ID
Color
C:\TEMP\CFD_標準教學系列_1_強制...
Analysis Case
step1 : Steady State CFD
step2 : Transient CFD
Solve
Export
Reload Result File
Copy
Delete
Rename
Model
LBC
Analysis & Results
Reload Result File
(可以先檢視分析結果)

分析結果
(最後1増量)

Steday CFD



Transient CFD



CFD_標準教學系列

強制對流分析10-2D斯特勞哈爾數

台灣邁達斯

註:範例所需計算時間較長

斯特勞哈爾數
(Strouhal number)

斯特勞哈爾數(Strouhal number) ，是流體力學中用來描述周期性非定常流動的一個無量綱數。

斯特勞哈爾數

$$\mathrm{St} = \frac{fL}{U}$$

St:斯特勞哈爾數
F:旋渦脫落頻率
L是特徵長度
U是流速

Reference
https://en.wikipedia.org/wiki/Strouhal_number

分析說明

Re:150 U:0.2 (m/s)

動力黏度計算(雷諾數:150)

層流Laminar Flow Re:150

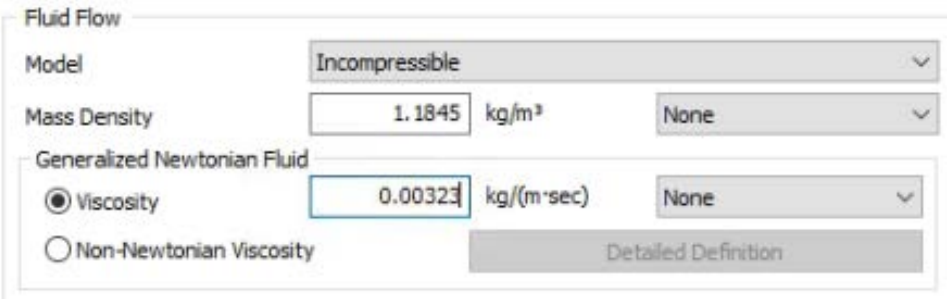
ρ:1.1845 (kg/m^3)=>(AIR_25℃)

U:0.2(m/s)

D:2.05(m)=>橋面截面積4.205m^2，特徵長度取面積開根號。

μ: ρ × U ×D/Re=0.00323 (kg/m-s)=>動力黏度

$$\mathrm{Re} = \frac{\rho V D}{\mu} = \frac{V D}{\nu} = \frac{Q D}{\nu A}$$



動能和渦長標度計算

Eddy Kinetic Energy(m^2/sec^2)

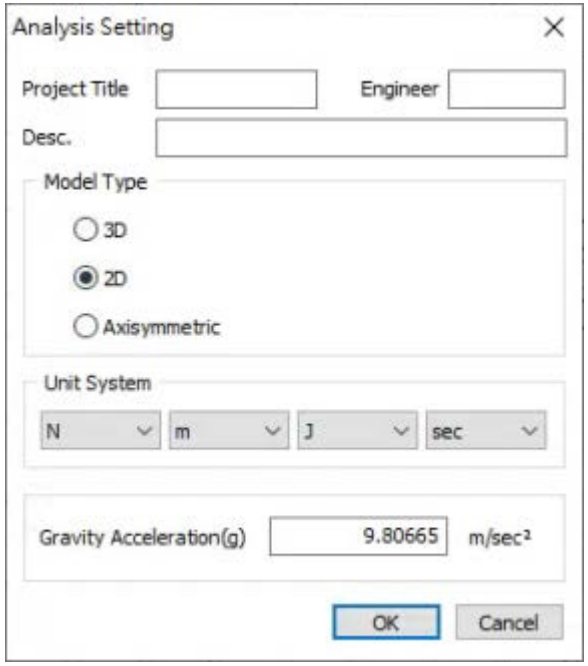
$$k = \frac{3}{2}(UI)^2 \qquad I = 0.16Re^{-\frac{1}{8}}$$

I :is the initial turbulence intensity [%]
U: is the initial velocity magnitude
Re=150
U=0.2(m/s)
$I=0.16 \times 150^{-1/8}=0.16 \times 0.5345=0.08552$
$K=(3/2) \times (0.2 \times 0.08552)^2= 0.0004388$

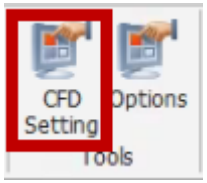
Eddy Length Scale or turbulent length scale

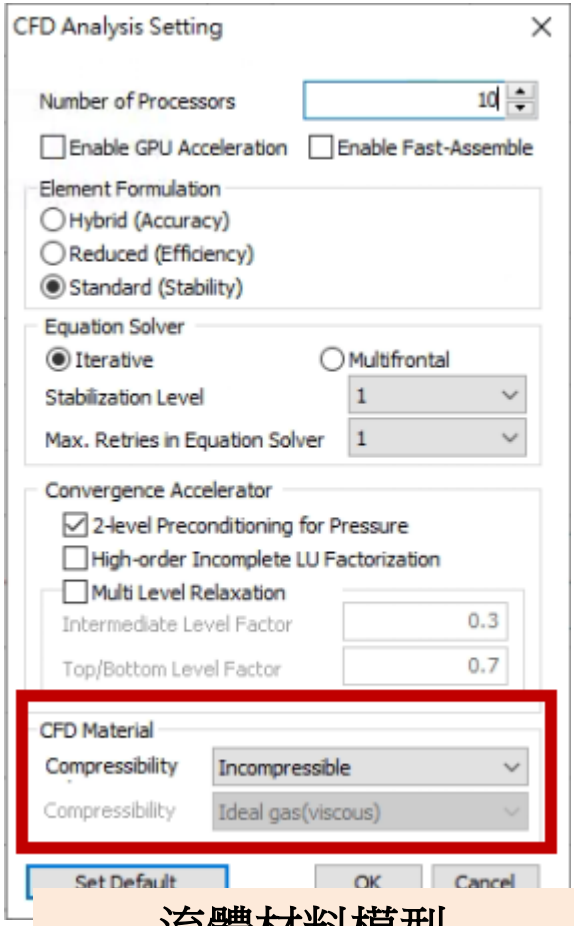
l:0.07L=0.07 ×2.05=0.1435
L:2.05(m) ， is a characteristic length.
(橋面截面積4.205m^2，特徵長度L取面積開根號)

環境



單位使用N/m/J/sec

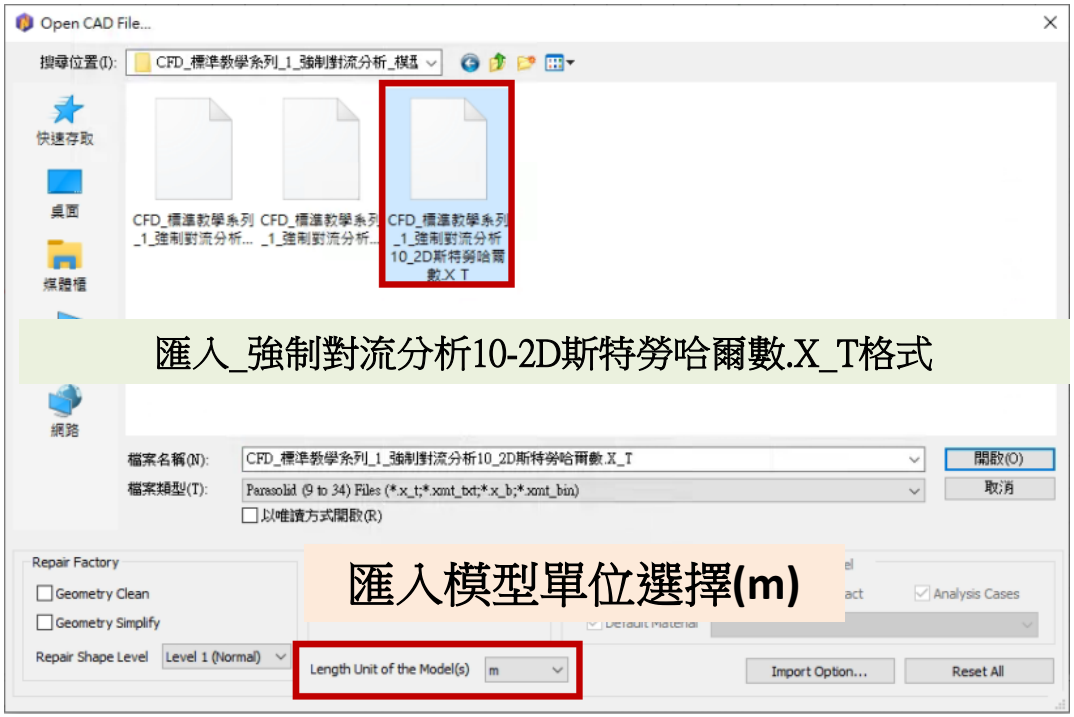


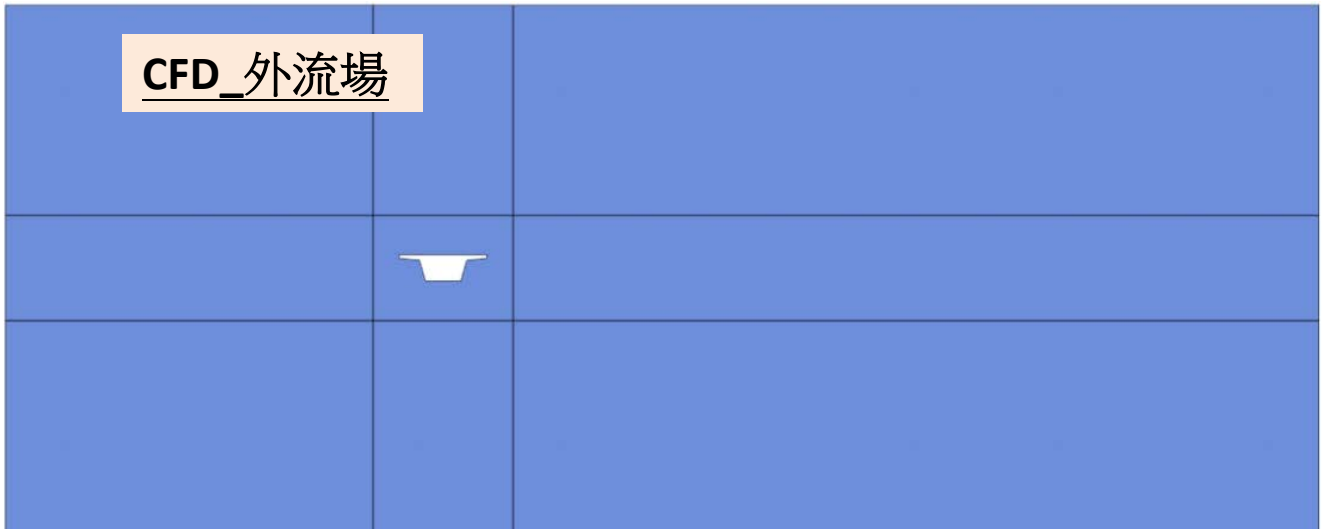


流體材料模型
不可壓縮流

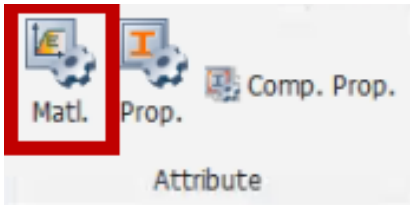
匯入模型

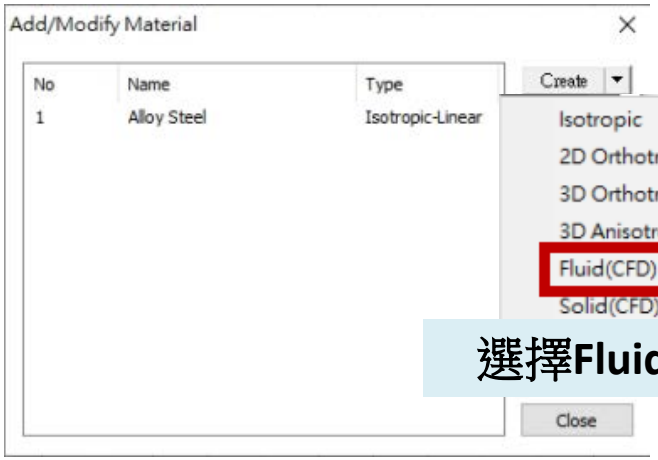

CAD
Import
CAD
Export
CAD File


Open CAD File...
搜尋位置(I):
CFD_標準教學系列_1_強制對流分析_模型
快速存取
桌面
媒體櫃
網路
CFD_標準教學系列_1_強制對流分析...
CFD_標準教學系列_1_強制對流分析...
CFD_標準教學系列_1_強制對流分析10_2D斯特勞哈爾數.X_T
檔案名稱(N):
CFD_標準教學系列_1_強制對流分析10_2D斯特勞哈爾數.X_T
檔案類型(T):
Parasolid (9 to 34) Files (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
以唯讀方式開啟(R)
開啟(O)
取消
匯入_強制對流分析10-2D斯特勞哈爾數.X_T格式
匯入模型單位選擇(m)
Repair Factory
Geometry Clean
Geometry Simplify
Repair Shape Level
Level 1 (Normal)
Length Unit of the Model(s)
m
Analysis Cases
Import Option...
Reset All


CFD_外流場

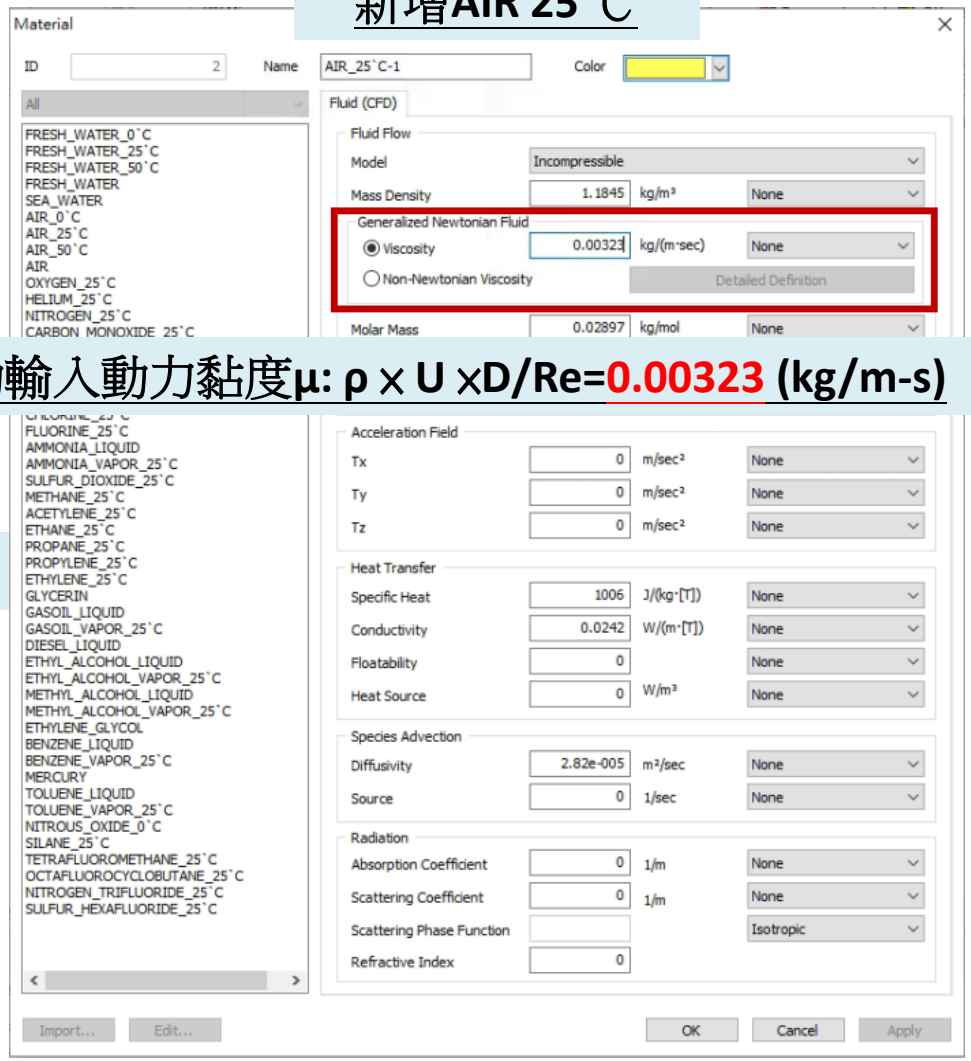
材料





選擇Fluid(CFD)

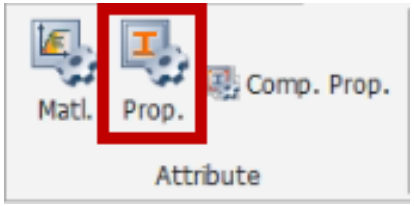
新增AIR 25 ℃



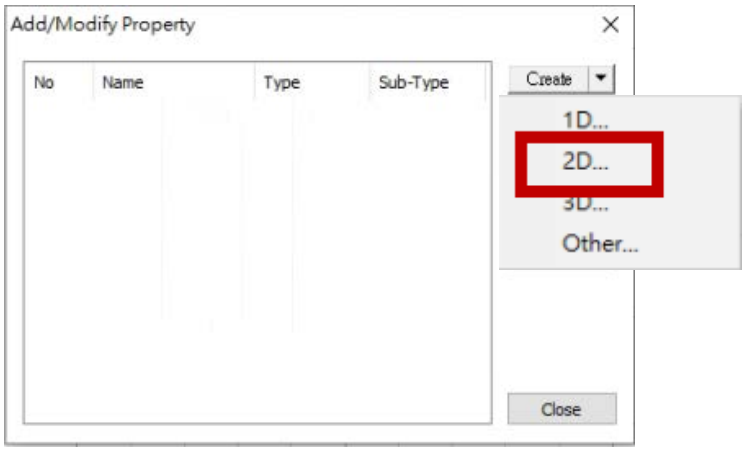
手動輸入動力黏度μ: ρ x U xD/Re=0.00323 (kg/m-s)

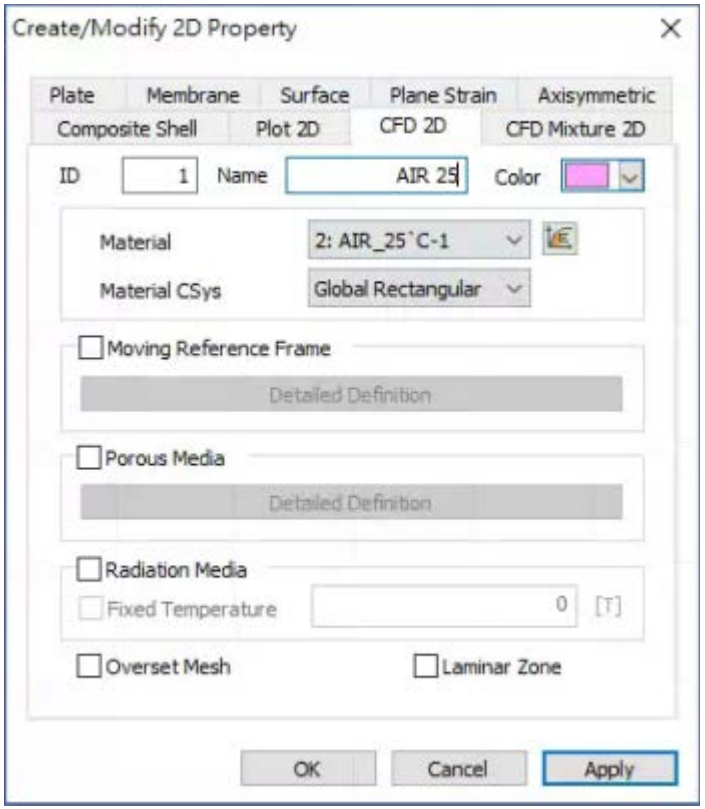
註:NFX CAE/CFD是整合性軟體,材料和屬性包含結構/熱/流場。

屬性



新增AIR 25 ℃ Property



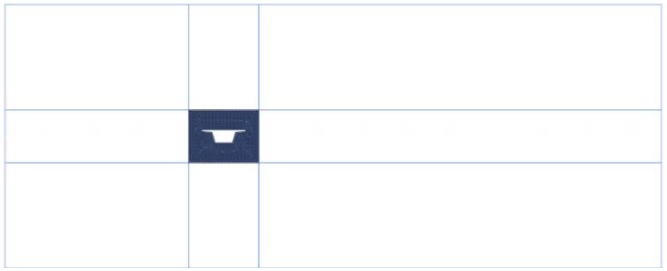


midas NFX-CFD

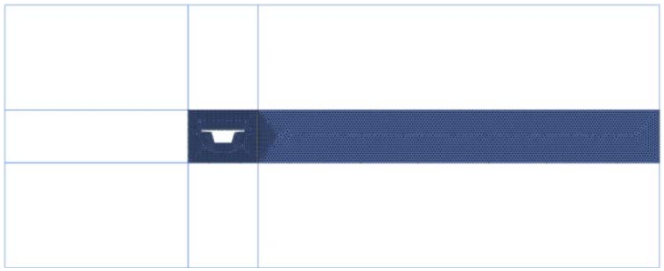
2D網格

三邊形網格/合併節點

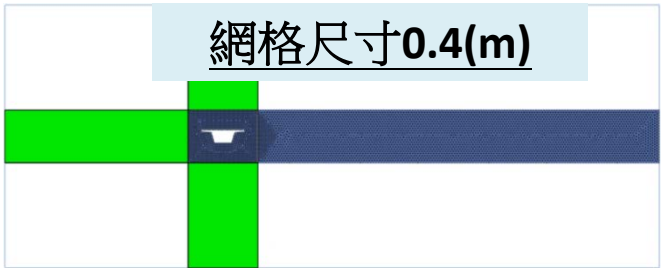
網格尺寸0.2(m)

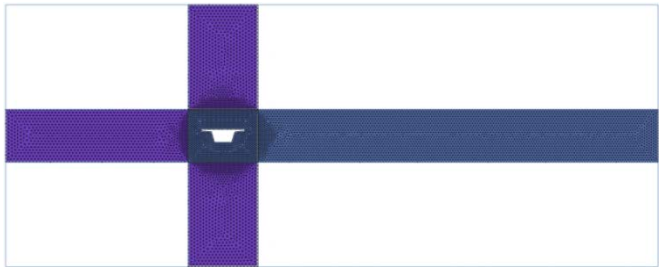


網格尺寸0.3(m)

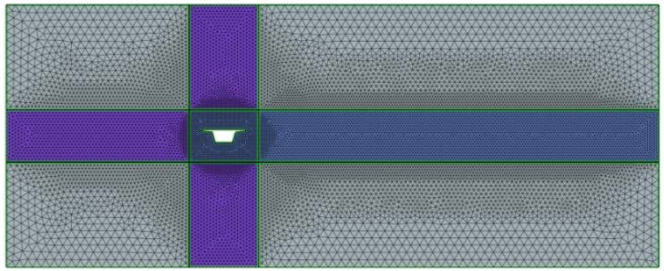


網格尺寸0.4(m)

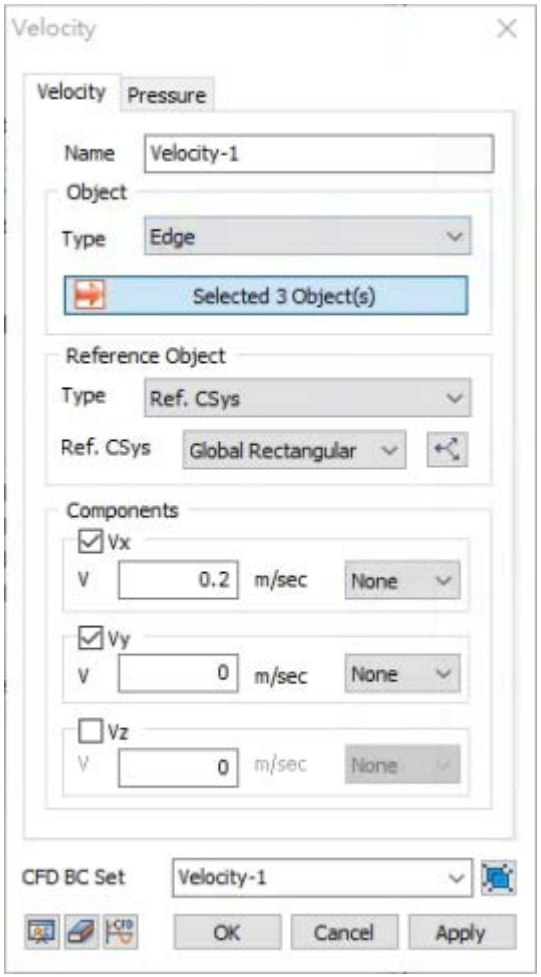




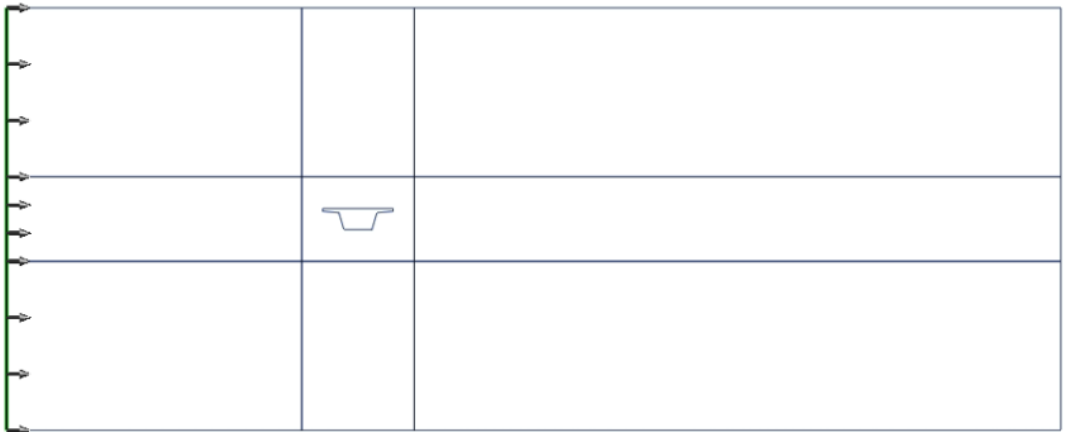
網格尺寸1(m)



入流

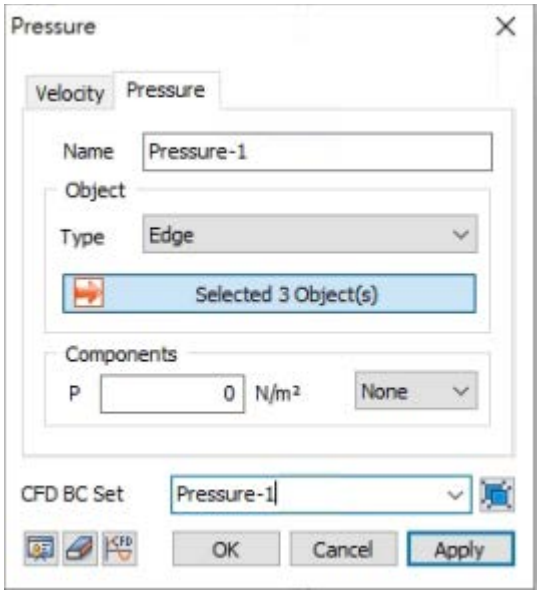


Vx=0.2 (m/s)　Vy=0 (m/s)

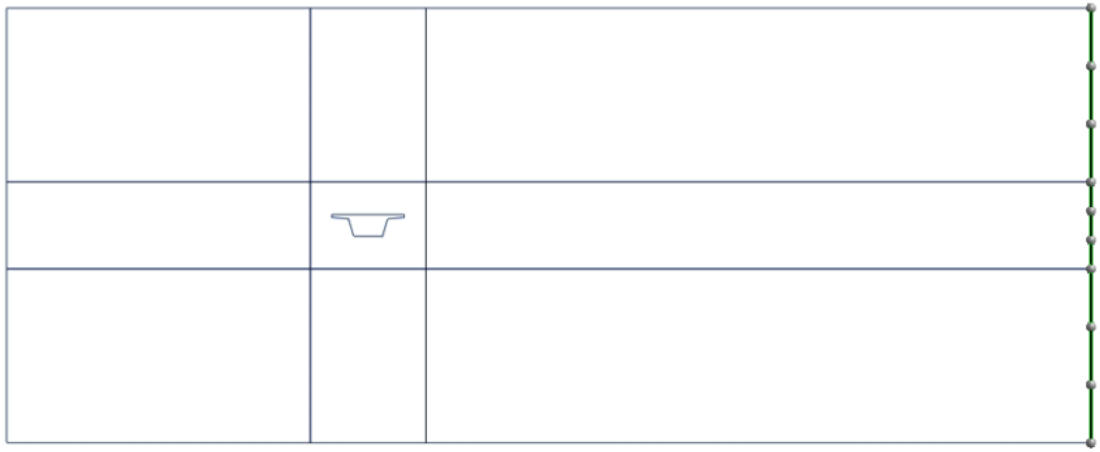


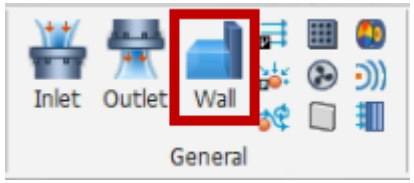
출류口


Inlet
Outlet
Wall
General


Pressure
Velocity
Pressure
Name
Pressure-1
Object
Type
Edge
Selected 3 Object(s)
Components
P
0
N/m²
None
CFD BC Set
Pressure-1
OK
Cancel
Apply

P=0 (N/m²)

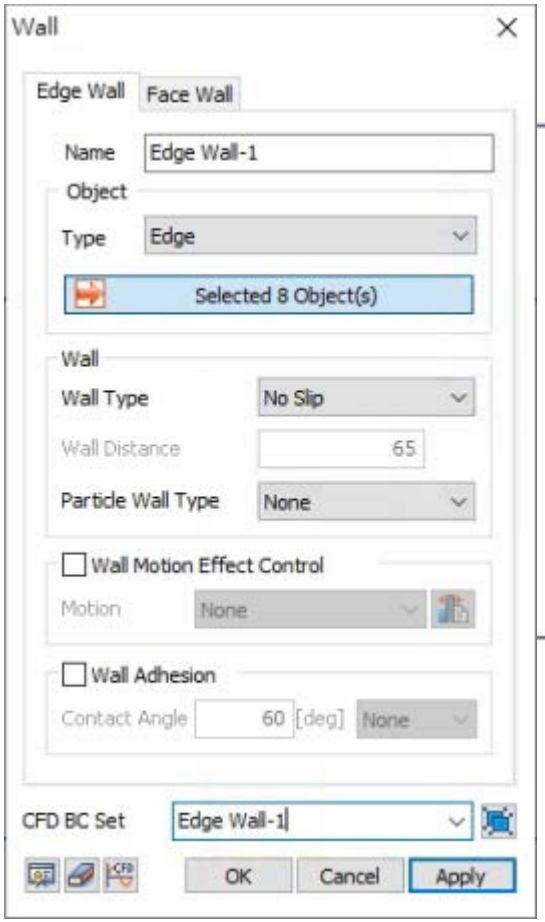


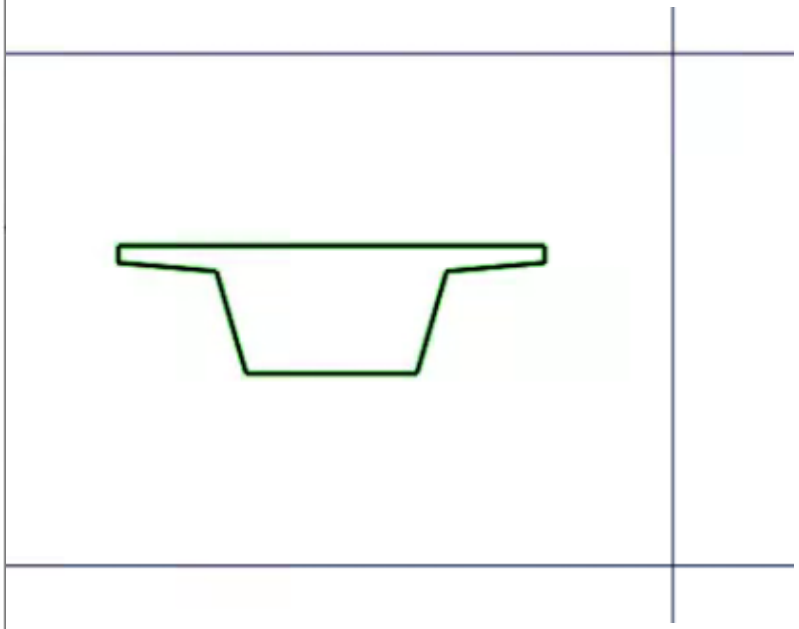


壁函數(No Slip)

No Slip(Wall Function)

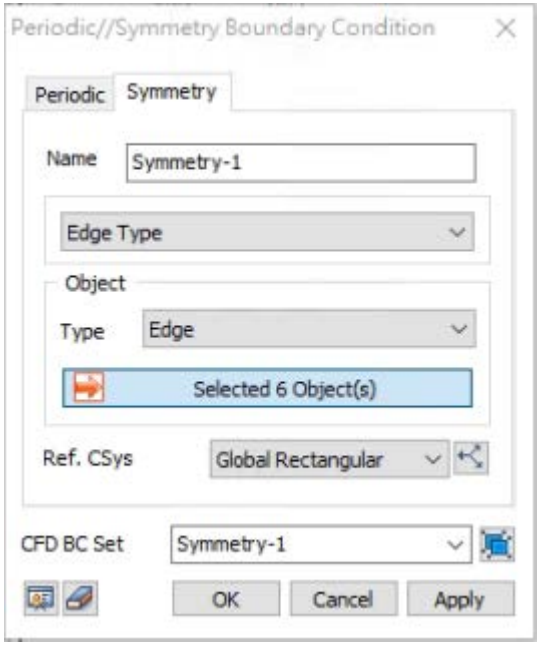
邊界處的速度為 0 , 適用於低或中雷諾數的問題。





對稱(Symmetry)


Inlet
Outlet
Wall
General


Periodic//Symmetry Boundary Condition
Periodic
Symmetry
Name
Symmetry-1
Edge Type
Object
Type
Edge
Selected 6 Object(s)
Ref. CSys
Global Rectangular
CFD BC Set
Symmetry-1
OK
Cancel
Apply

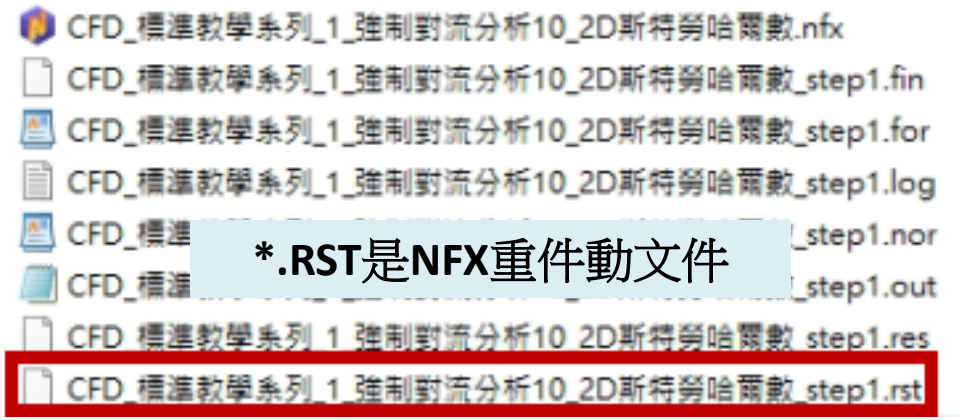
分析說明
(重啓動)

Solution Type : Step1(Steady CFD)

自動寫出重啓動文件

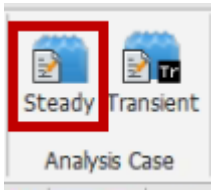
Solution Type : Step2(Transient CFD)

匯入重啓動文件

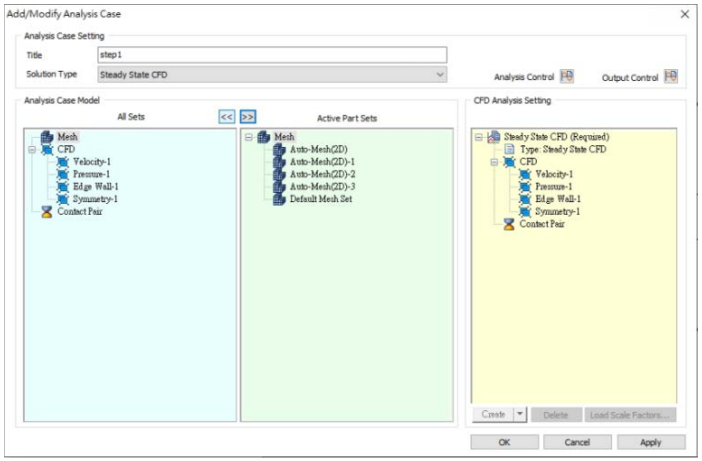


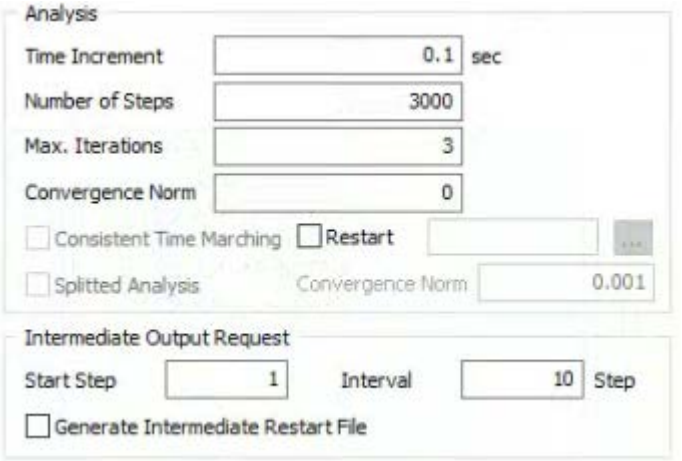
***.RST是NFX重件動文件**

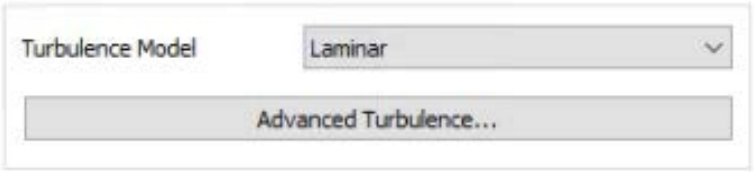
Step1 CFD穩態分析

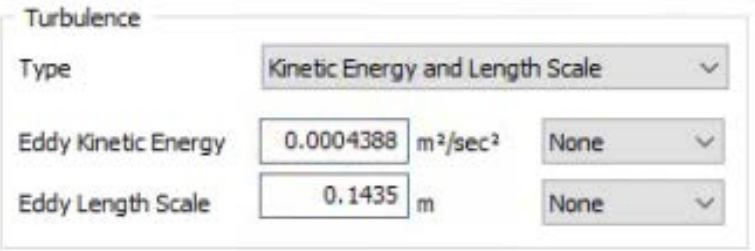


Solution Type : Steady State CFD







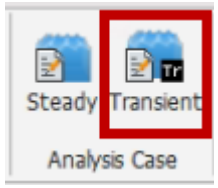


Eddy Kinetic Energy(m^2/sec^2)

K=0.0004388

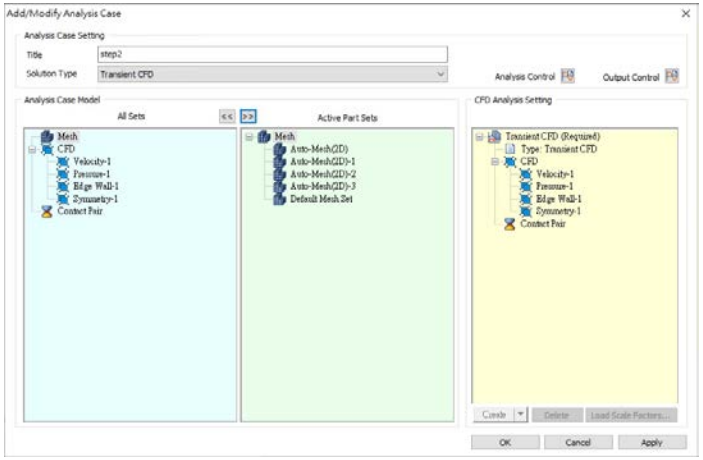
Eddy Length Scale or turbulent length scale

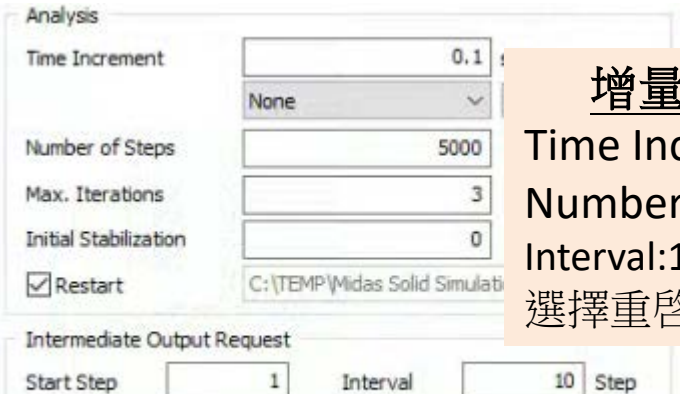
L=0.1435



Step2 CFD瞬態分析

Solution Type : Transient CFD





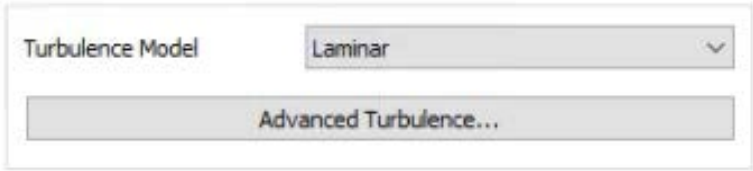
增量步和收斂定義

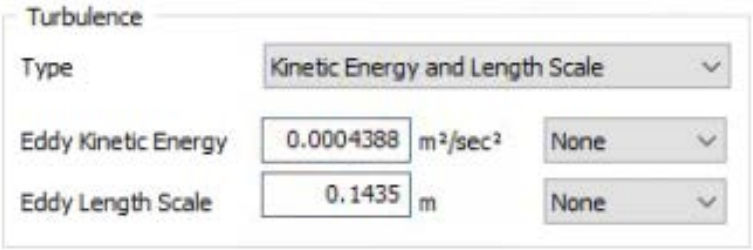
Time Increment :0.1

Number of steps:5000

Interval:10

選擇重啓動文件檔Step1.rst





Eddy Kinetic Energy(m^2/sec^2)

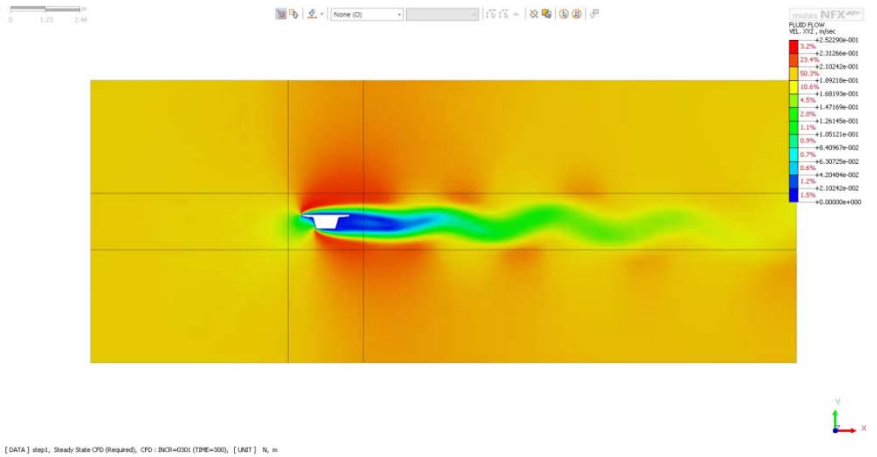
K=0.0004388

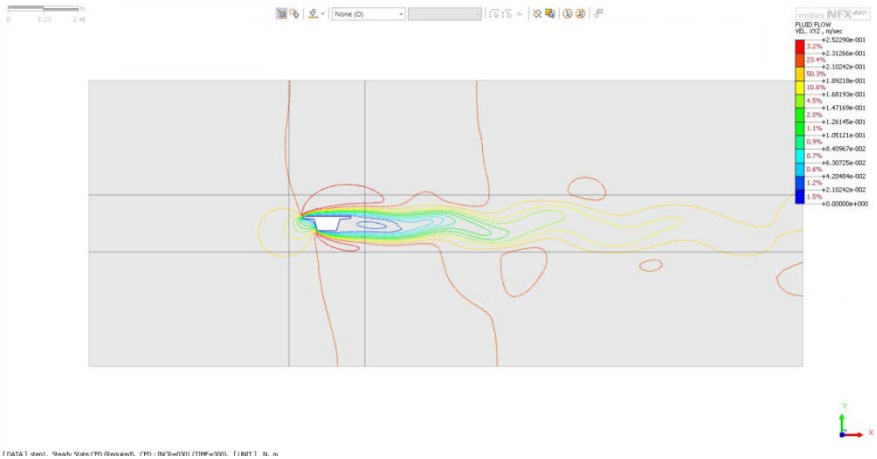
Eddy Length Scale or turbulent length scale

L=0.1435

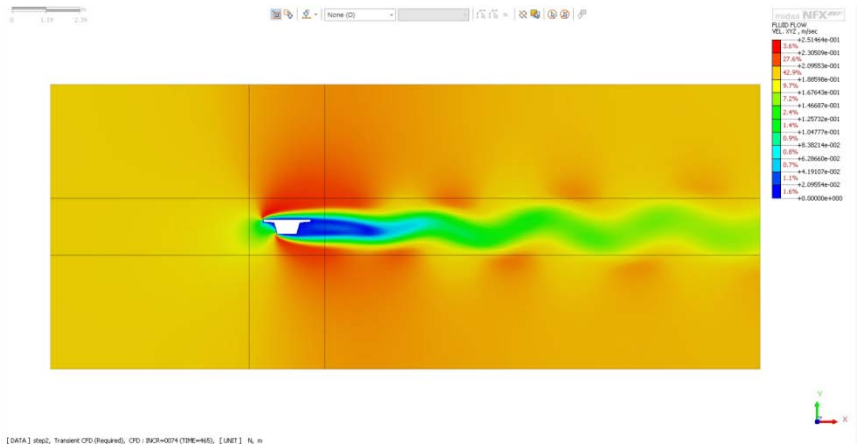
분석결과

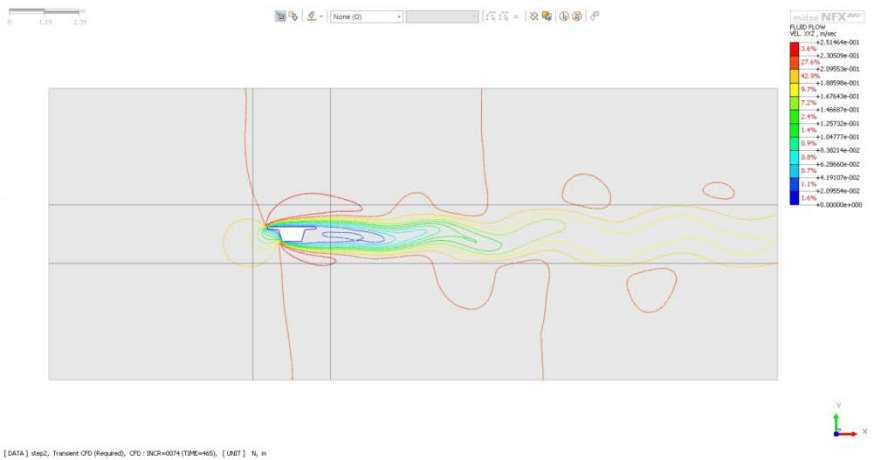
Steday CFD



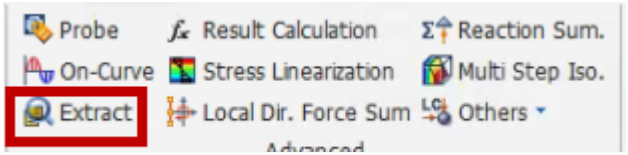


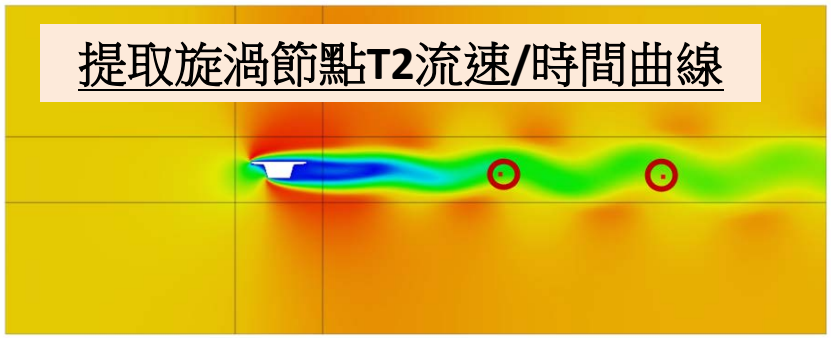
Transient CFD





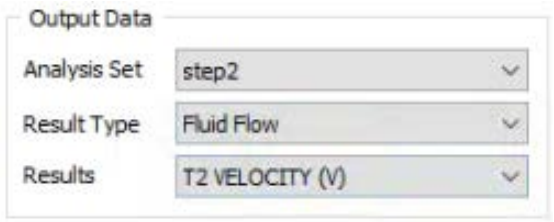
旋渦脫落頻率

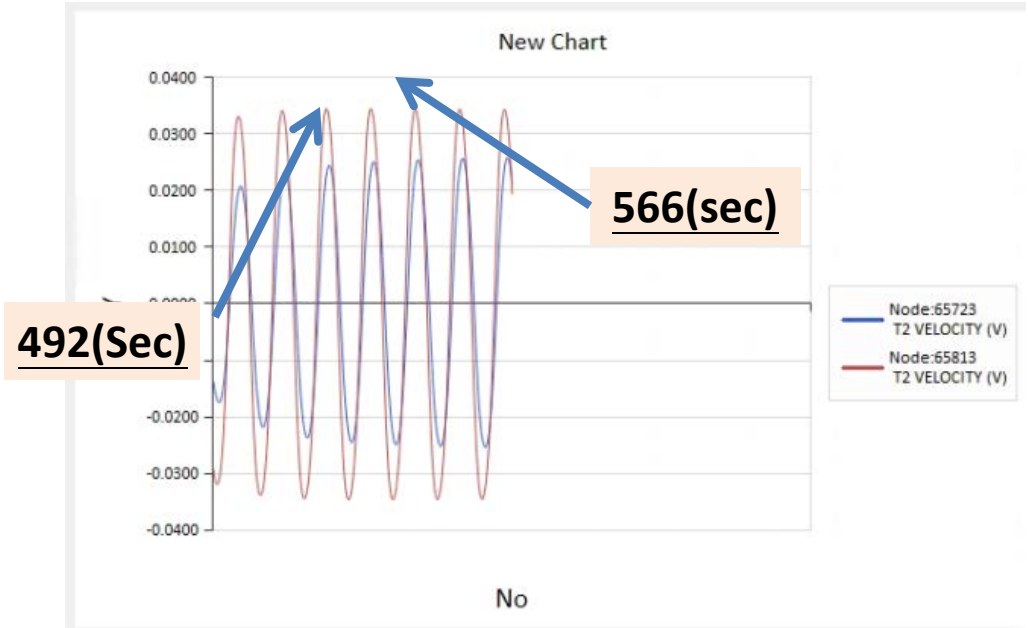




旋渦脫落頻率

Freq=1/(566-492)=0.0135Hz





斯特勞哈爾數
(Strouhal number)

斯特勞哈爾數

$$\mathrm{St} = \frac{fL}{U}$$

St:0.138375=>斯特勞哈爾數
f:0.0135(Hz)=>旋渦脫落頻率
L:2.05(m)=>橋面截面積4.205m^2，特徵長度取面積開根號。
U:0.2(m/sec)=>流速

midas NFX-CFD

CFD_標準教學系列

強制對流分析11-2D風廓線

台灣邁達斯

風廓線
(Wind Speed Profile)

風廓線即風速隨高度的變化曲線，以研究大氣邊界層內的風速規律。常用邊界層風速剖面的有兩定律，一為指數律(power law)，另一則為對數律(logarithmic law)。

指數律風廓線
(Wind profile power law)

$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

u:is the wind speed(m/sec) at height z (m)
u_γ:is the known wind speed(m/sec) at a reference height z_γ (m)
α : is an empirically derived coefficient that varies dependent upon the stability of the atmosphere.For neutral stability conditions , α is approximately 1/7 。

對數風廓線
(Log wind profile)

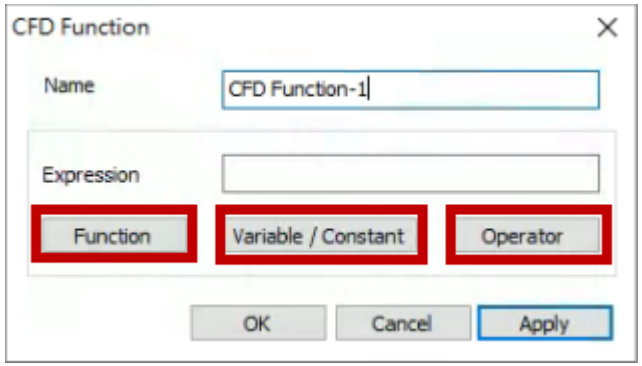
$$u_z = \frac{u_*}{\kappa} \left[\ln \left(\frac{z-d}{z_0} \right) \right]$$

u_z:is the wind speed(m/sec) at height z (m)
u_* : is the friction velocity (m/s)
k: is the Von Kármán constant (~0.41)
d : is the zero plane displacement (m)
Z_0:is the surface roughness (m)

Reference
https://en.wikipedia.org/wiki/Wind_profile_power_law
https://en.wikipedia.org/wiki/Log_wind_profile

CFD函數

NFX CFD提供複雜函數和矩陣計算，它還包括一些幫助插入函數的工具：變量列表和運算符列表。



風廓線CFD函數定義
(指數律風廓線)

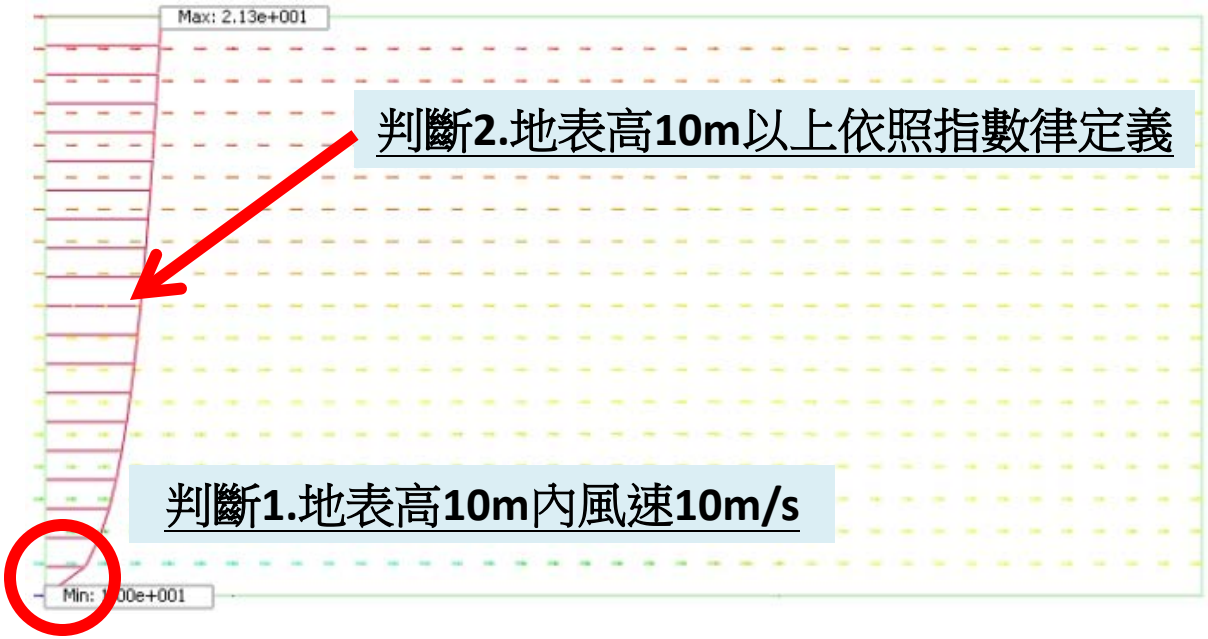
$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

u:相對高度風速(m/s)
Z_γ:地表高度(10m)
u_γ:地表高度Z_γ風速(假設10m/s)
α :0.143

CFD IF Function格式

if(expression)then(elif_expression)else(next_expression)endif

=>if(y<10)then(10)else(10*(y/10)^(0.143))endif



註:2D空間高度座標為y軸。

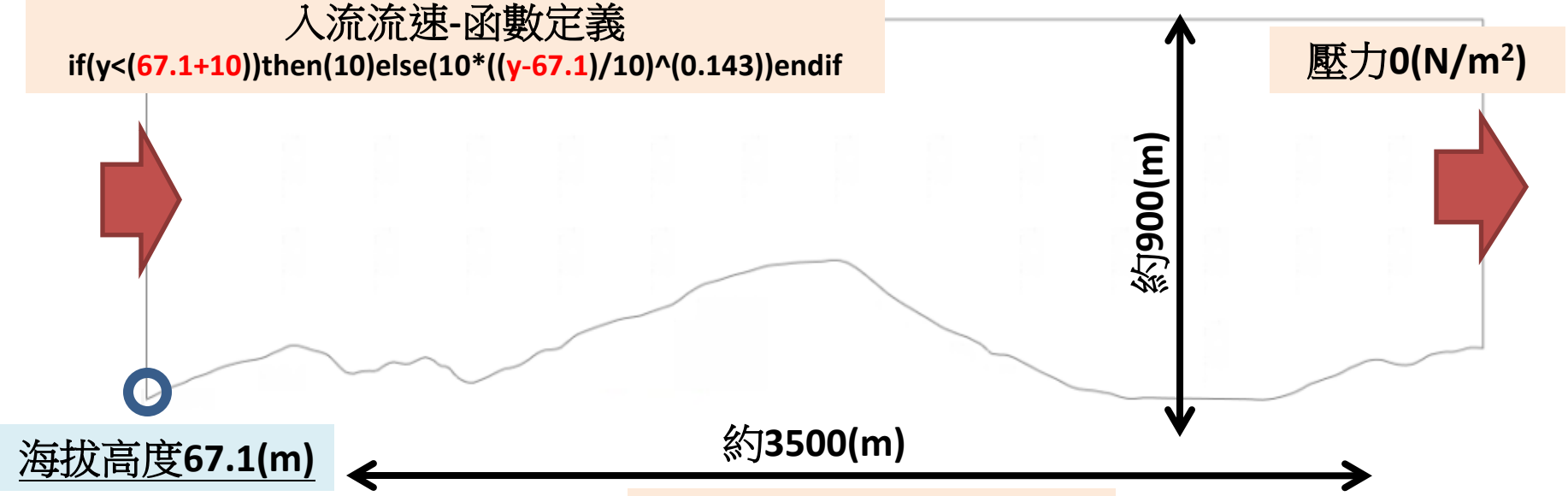
分析說明
(地形風場分析)

AIR 25 ℃

空間特徵-對稱

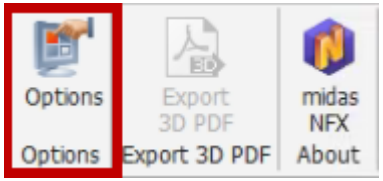
入流流速-函數定義
if(y<(67.1+10))then(10)else(10*((y-67.1)/10)^(0.143))endif

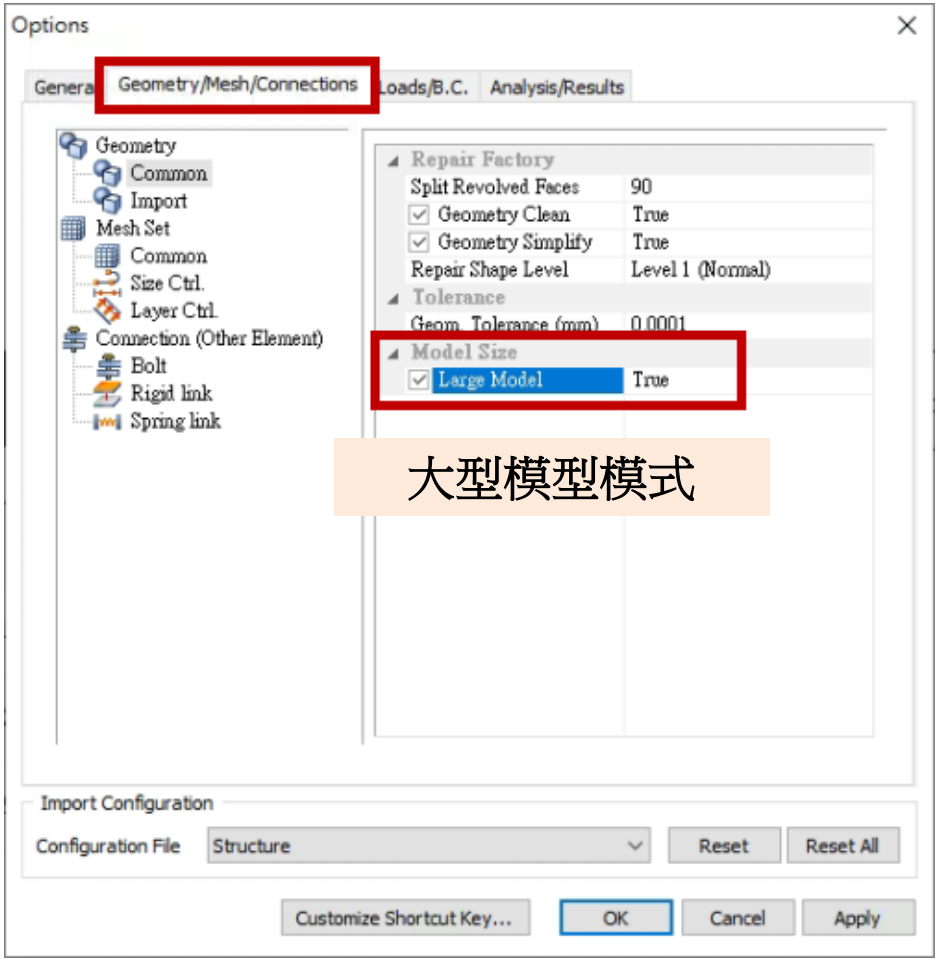
壓力0(N/m²)



地表壁函數
Dimensionless Wall y+=65

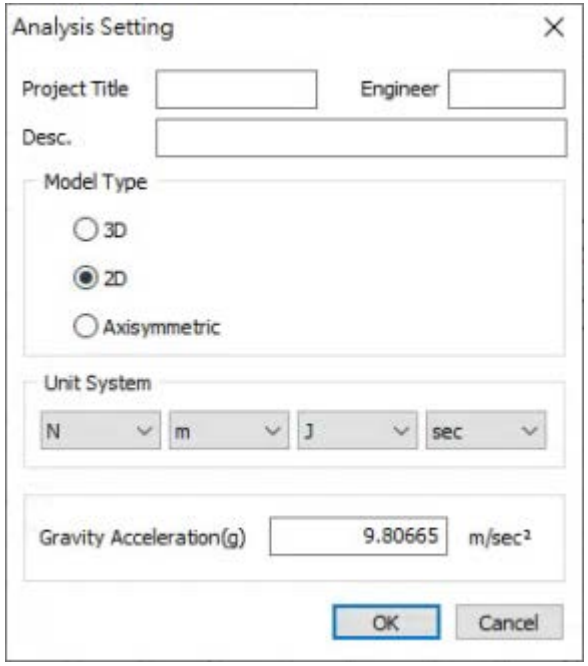
環境-1



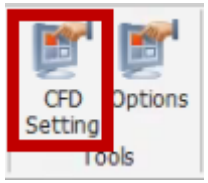


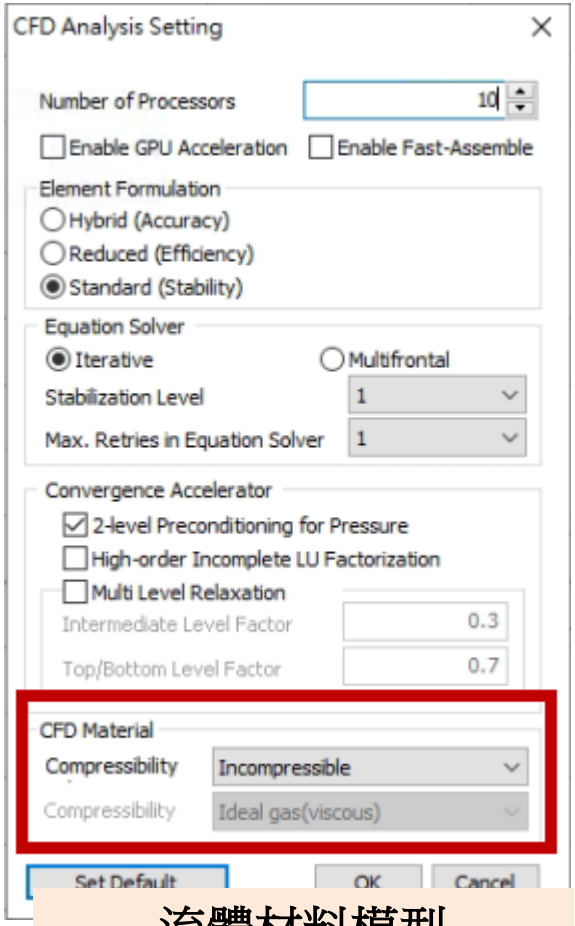
註:標準模型模式1000m ×1000m ×1000m

環境-2



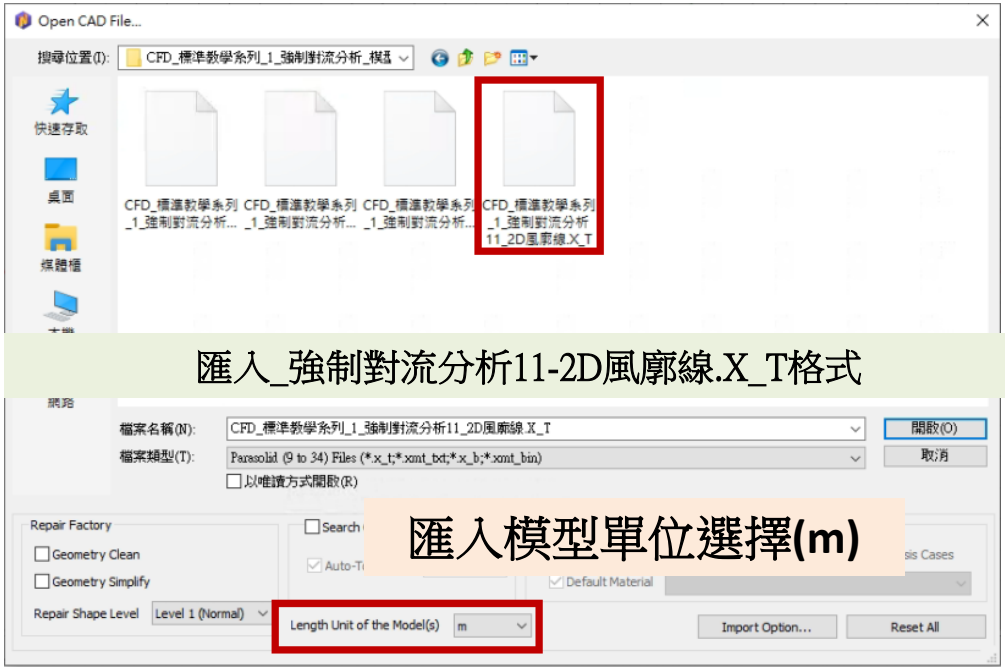
單位使用N/m/J/sec





流體材料模型
不可壓縮流

匯入模型

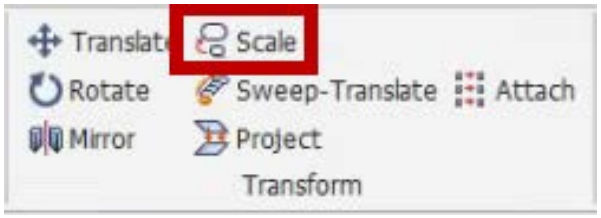


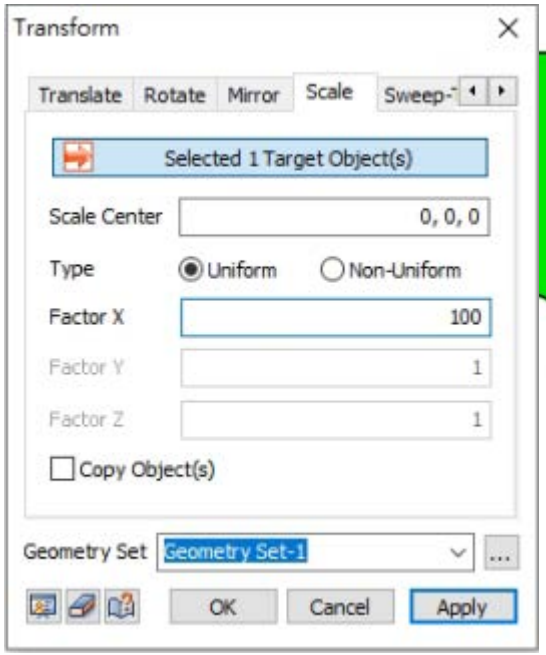
匯入_強制對流分析11-2D風廓線.X_T格式

匯入模型單位選擇(m)

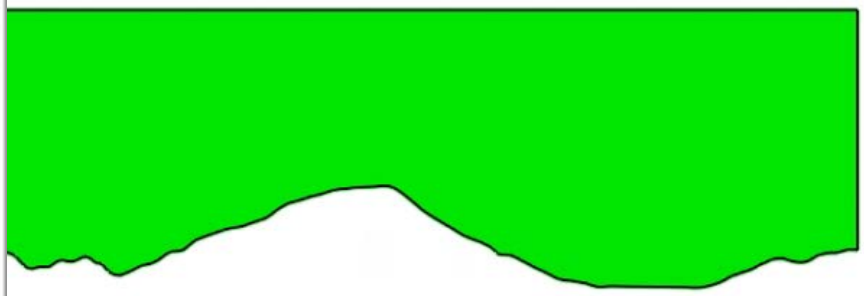
空間地型流場區域

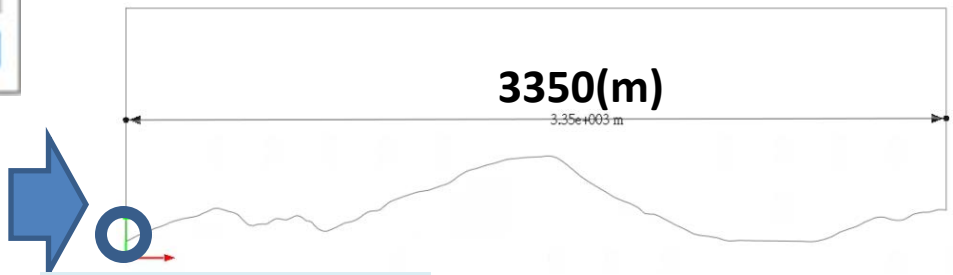
模型放大





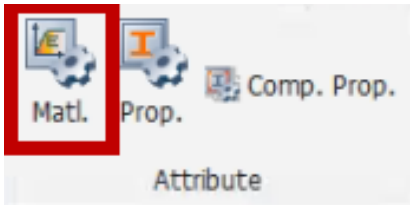
模型放大100倍

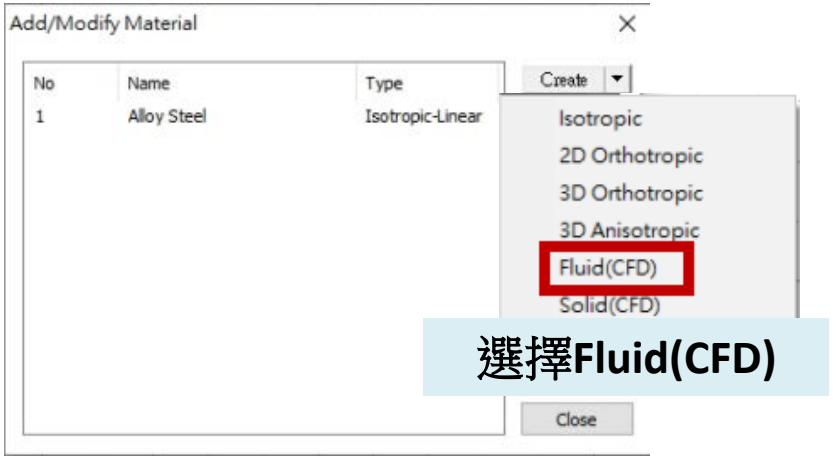




海拔高度67.1(m)

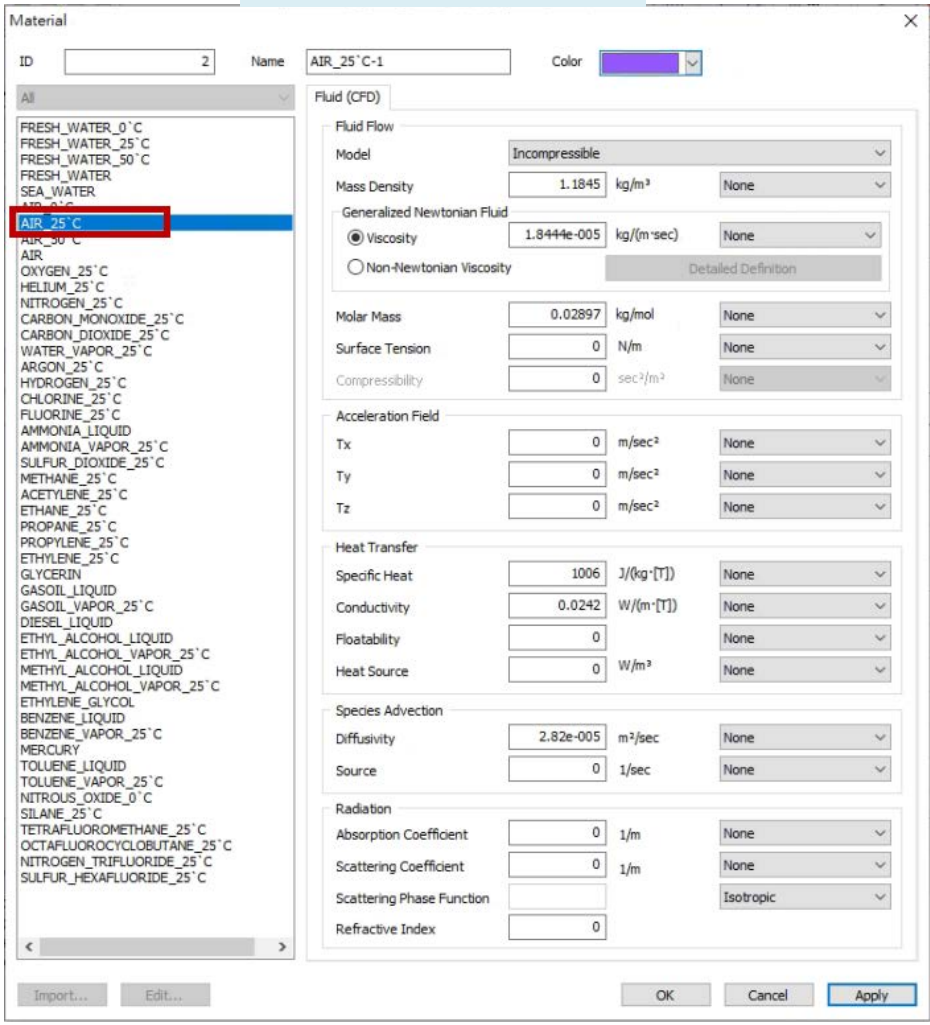
材料





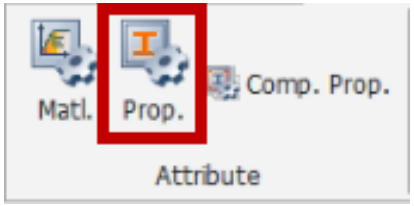
選擇Fluid(CFD)

新增AIR 25 ℃

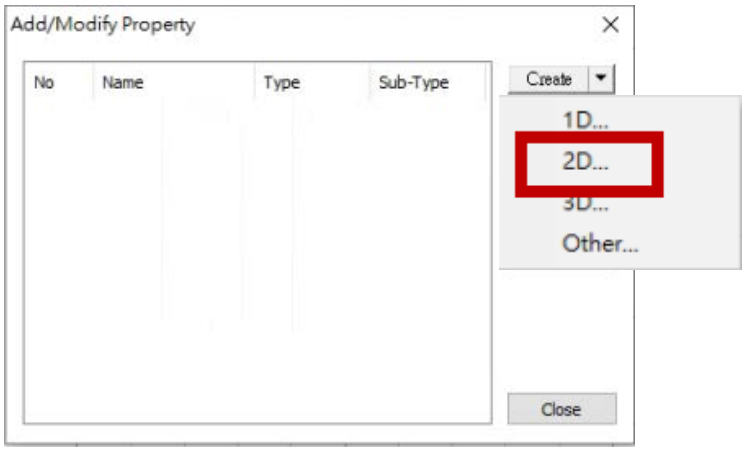


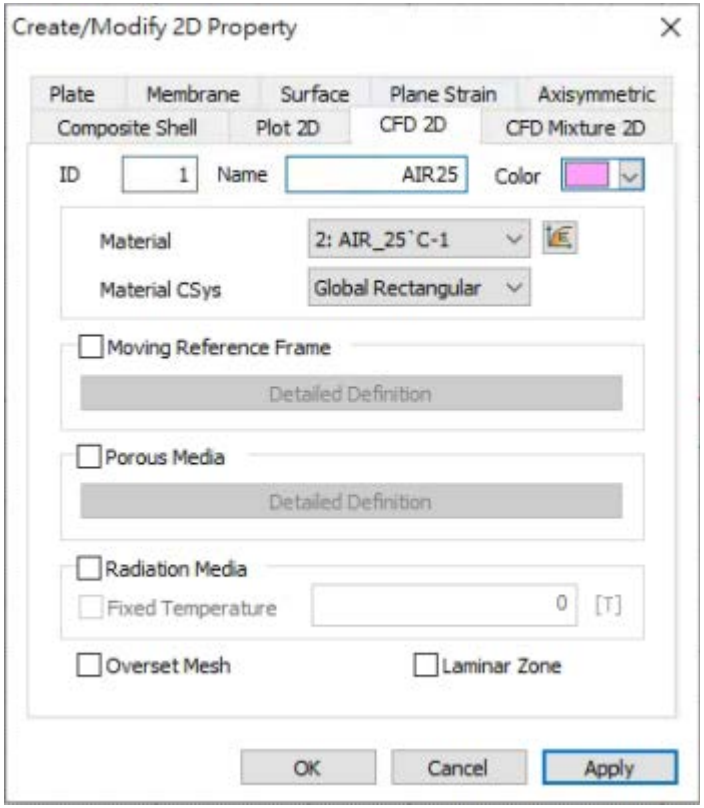
註:NFX CAE/CFD是整合性軟體,材料和屬性包含結構/熱/流場。

屬性

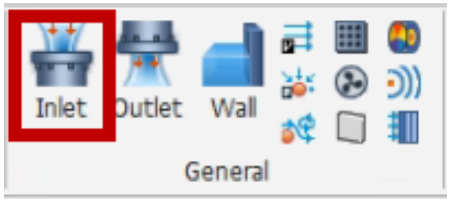


新增AIR 25 ℃ Property



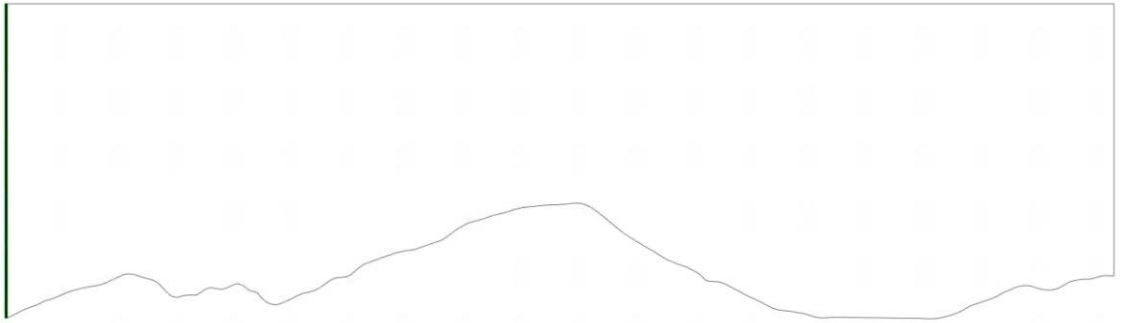


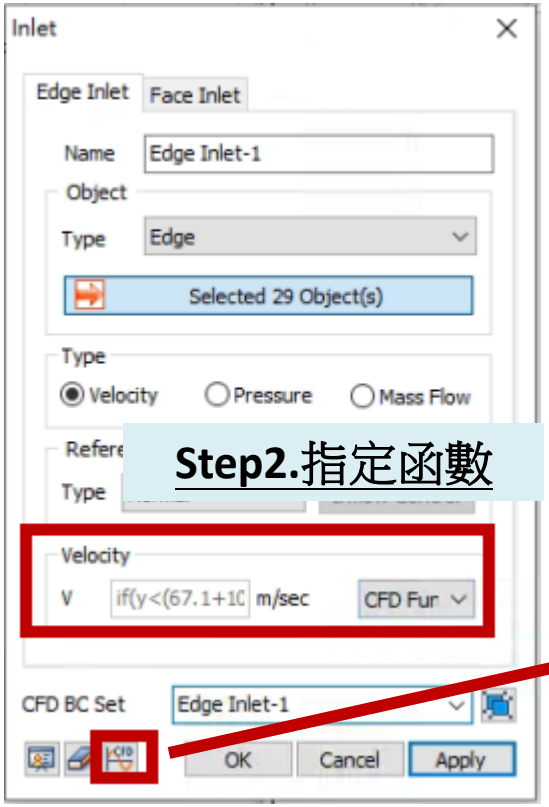
入流



入流流速-函數定義

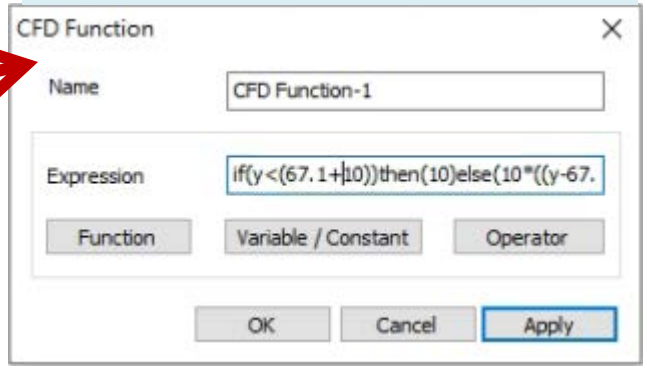
if(y<(67.1+10))then(10)else(10*((y-67.1)/10)^(0.143))endif





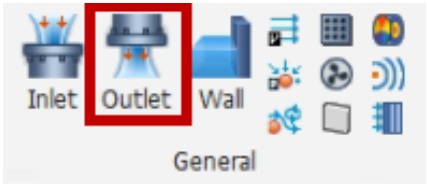
Step2.指定函數

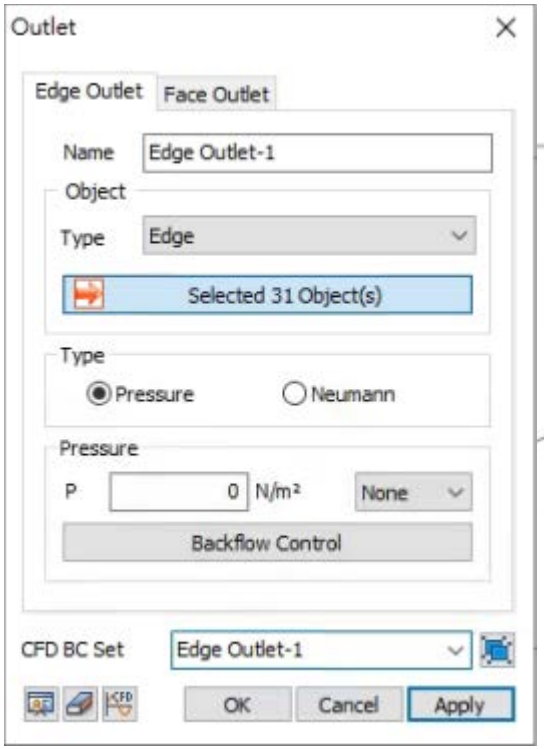
Step1.新增風廓線入流函數



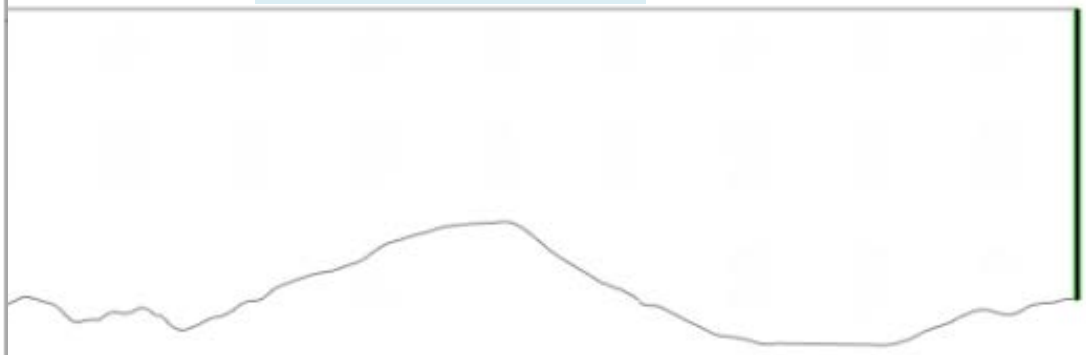
註:不同邊界&載荷建議使用不同集合名稱。

出流



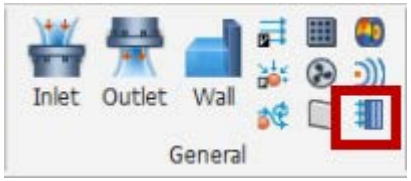


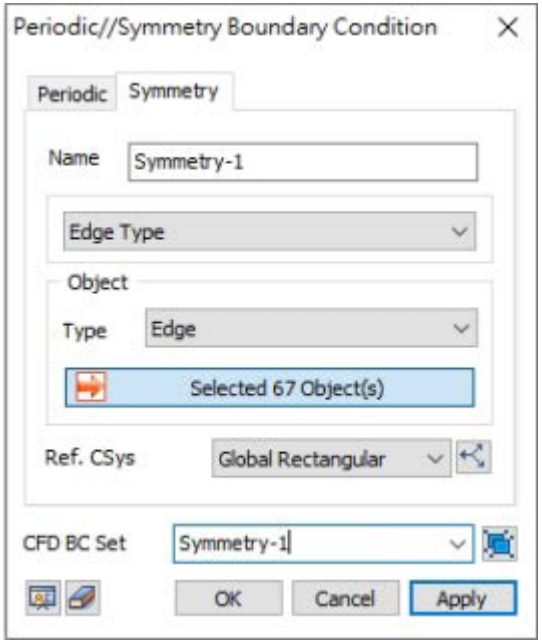
壓力0(N/m²)

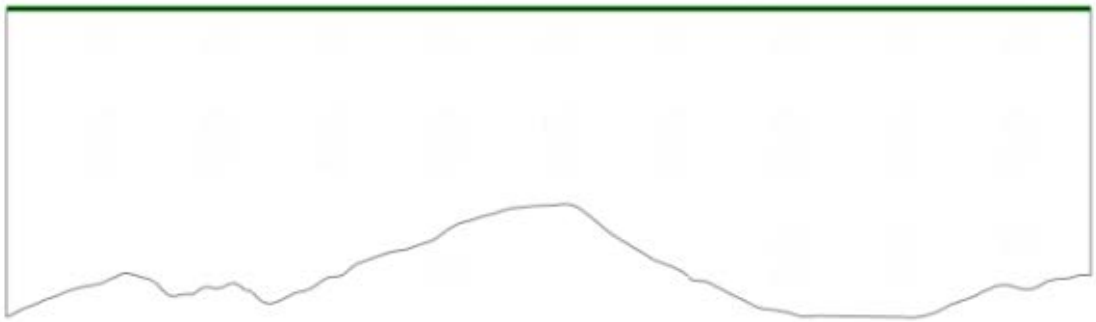


註:不同邊界&載荷建議使用不同集合名稱。

對稱

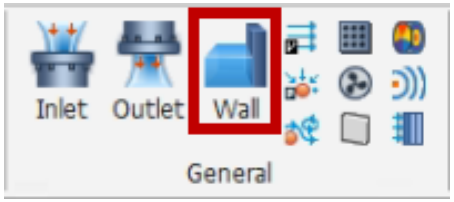


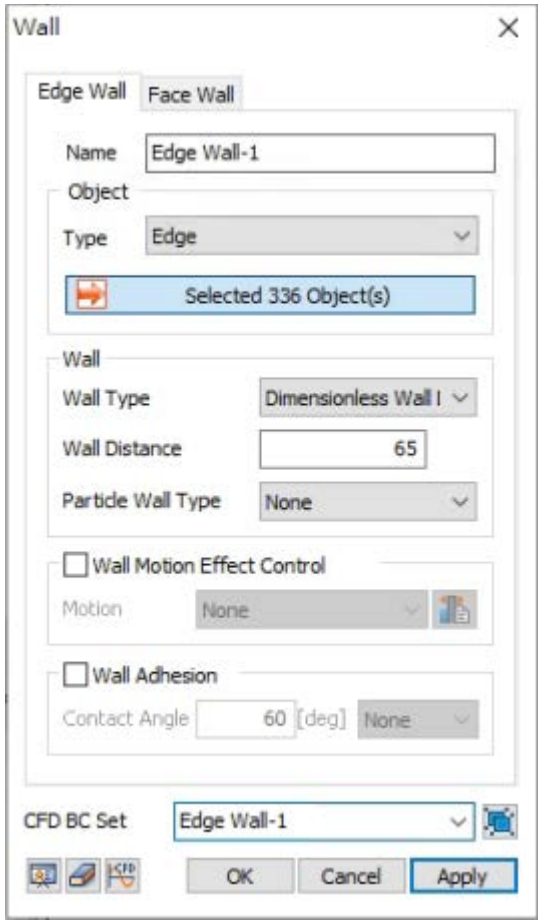




註:不同邊界&載荷建議使用不同集合名稱。

壁



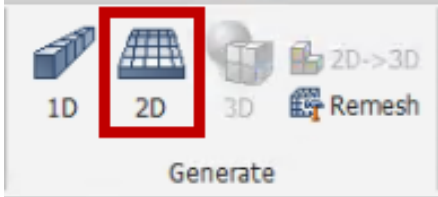


選取地表線特徵

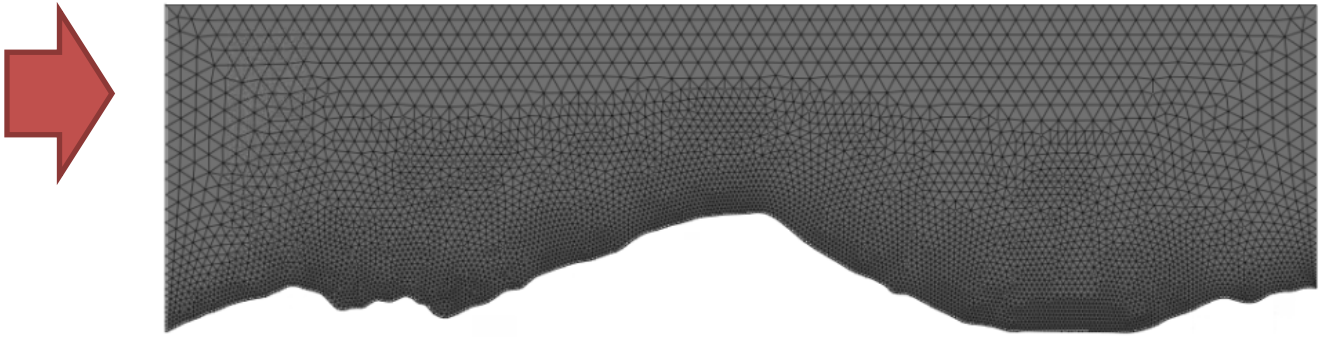
地表壁函數使用預設
Dimensionless Wall y+=65

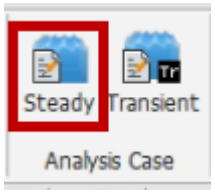
註:不同邊界&載荷建議使用不同集合名稱。

2D網格



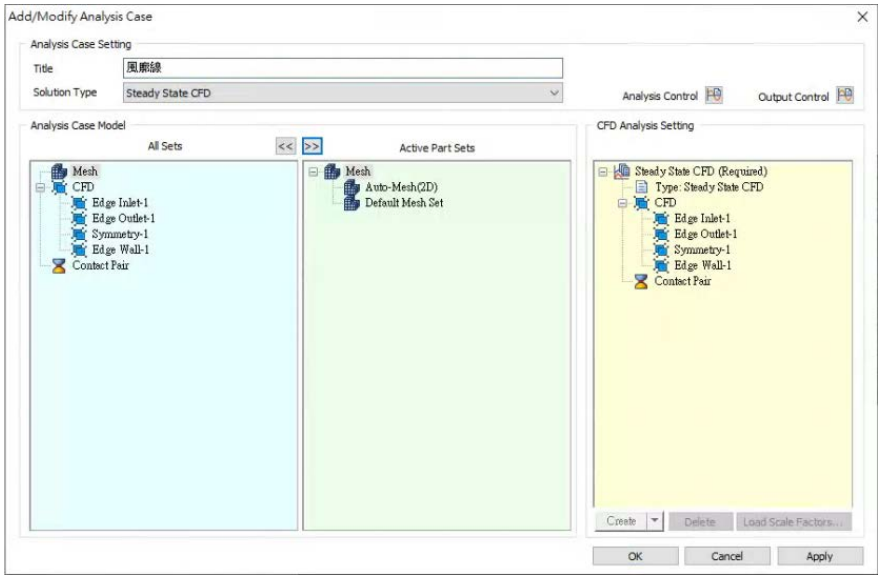
三邊形網格/合併節點/網格尺寸40(m)

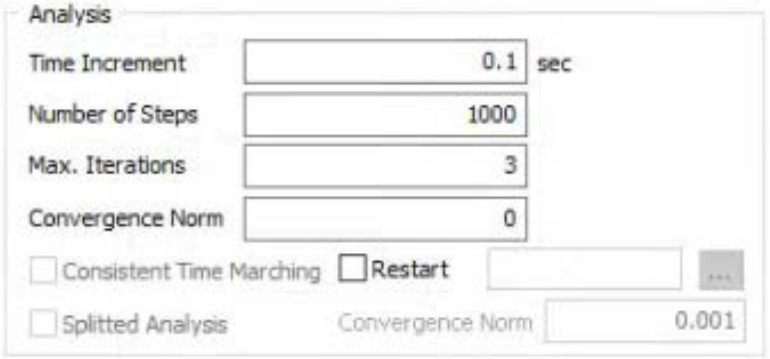


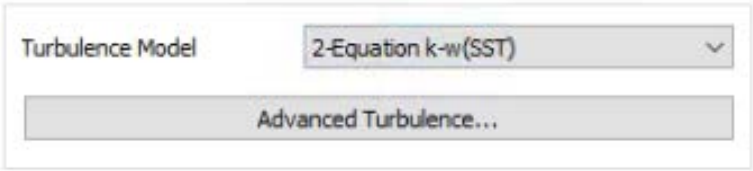


CFD穩態分析
(自訂函數-風廓線)

Solution Type : Steady State CFD

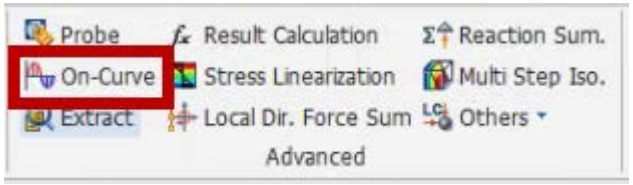


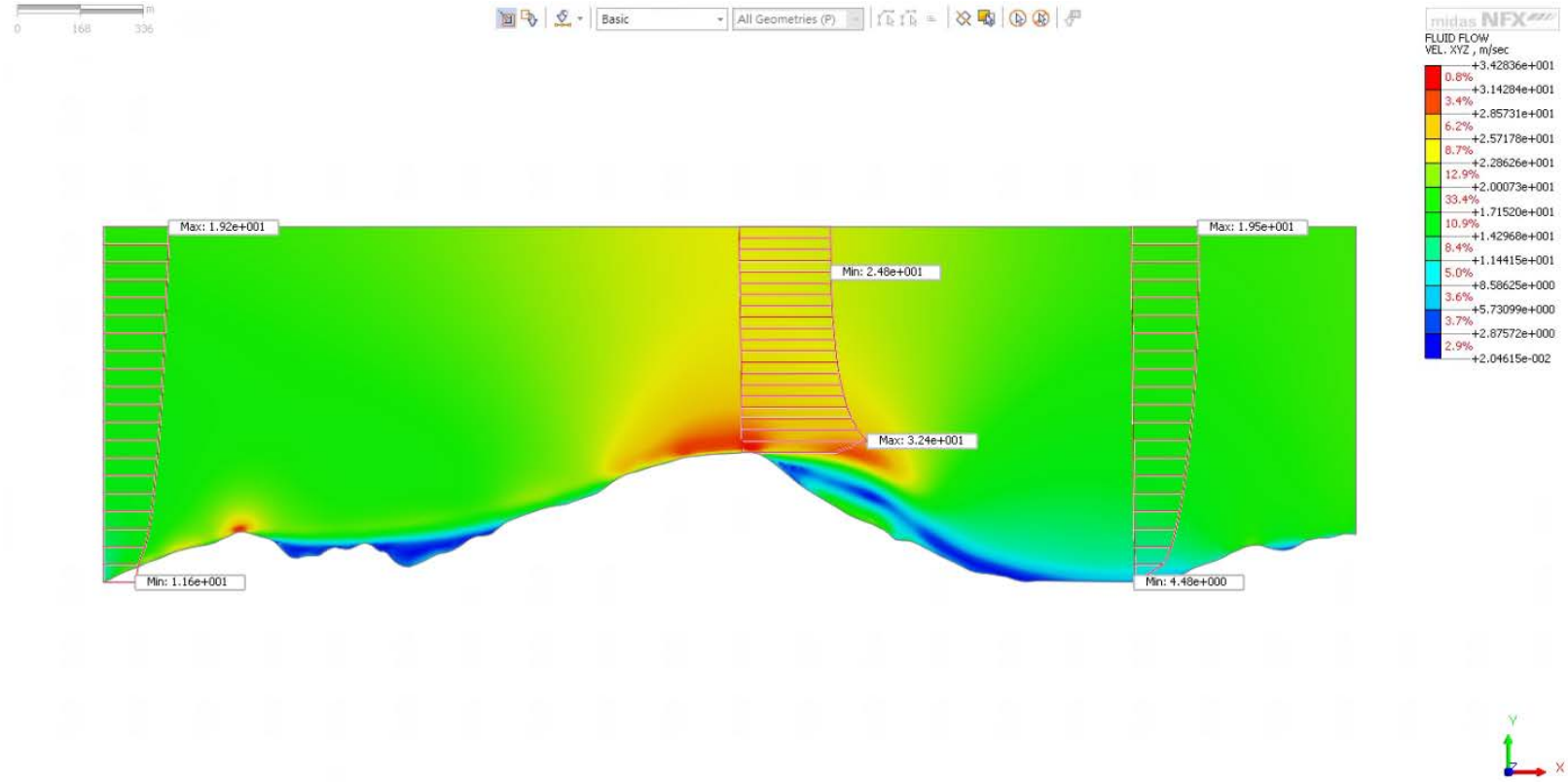




分析結果

(最後1増量)


Probe
Result Calculation
Reaction Sum.
On-Curve
Stress Linearization
Multi Step Iso.
Extract
Local Dir. Force Sum
Others
Advanced


Basic
All Geometries (P)
midas NFX
FLUID FLOW
VEL. XYZ , m/sec
+3.42836e+001
0.8%
+3.14284e+001
3.4%
+2.85731e+001
6.2%
+2.57178e+001
8.7%
+2.28626e+001
12.9%
+2.00073e+001
33.4%
+1.71520e+001
10.9%
+1.42968e+001
8.4%
+1.14415e+001
5.0%
+8.58625e+000
3.6%
+5.73099e+000
3.7%
+2.87572e+000
2.9%
+2.04615e-002
Max: 1.92e+001
Min: 2.48e+001
Max: 1.95e+001
Max: 3.24e+001
Min: 1.16e+001
Min: 4.48e+000
Y
Z
X
[DATA] 風圧線, Steady State CFD (Required), CFD : INCR=0101 (TIME=100), [UNIT] N, m

CFD_標準教學系列

強制對流分析13-2D壓力係數

台灣邁達斯

註:範例所需計算時間較長

壓力係數
(Pressure Coefficient)

壓力係數是描述流體動力學中整個流場的相對壓力的無量綱數，壓力系數應用於空氣動力學和流體力學，流體流場中的每一點的壓力系數不同。該無量綱係數與維數關係如下：

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty V_\infty^2} = \frac{p - p_\infty}{p_0 - p_\infty}$$

P: is the static pressure at the point at which pressure coefficient is being evaluated.

P_∞ : is the static pressure in the free-stream.

P_0: is the stagnation pressure in the free-stream.

ρ_∞ : is the free-stream fluid density.

V_∞ :is the free-stream velocity of the fluid, or the velocity of the body through the fluid.

The free-stream(自由流)

自由流是指飛機前未經擾動的來流，也即沒有飛機等干擾時，空氣的自然流動現象。

Reference

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

CAARC建築風場數值模擬-1

$$C_P = 2(P - P_0)/\rho U_0^2$$

C_p:風壓係數
P:觀測點平均壓力(通用標準測點取2/3H)
P_0:參考高度處的靜壓
ρ：空氣密度
U_0:建築物高度初始風速
H:建築物高度(m)

計算流域取1800m×600m×1000m
(建築物置於流域沿流向前1/3處)

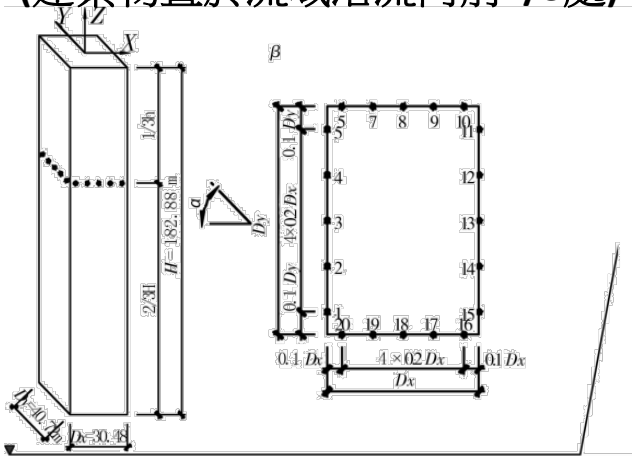


图 1　CAARC 几何尺寸及压力测点布置图

Reference

聶少鋒,周緒紅.CAARC標準高層建築三維鈍體繞流風場數值模擬.土木建築與環境工程,2009,Vol.31 No.6

CAARC建築風場數值模擬-2

$$U = U_0 \left(\frac{Z}{Z_0}\right)^{\alpha}$$

U:相對高度風速(m/s)
Z:地表相對高度(m)
Z_0:參考高度(常見取地表10m高)
U_0:參考高度風速(m/s)
α :地面粗糙度指數

类别	描述	Z_G/m	α
A	指近海海面、海岛、海岸、湖岸及沙漠地区	300	0.12
B	指田野、乡村、丛林、丘陵及房屋比较稀疏的乡镇和城市郊区	350	0.16
C	指有密集建筑群的城市市区	400	0.22
D	指有密集建筑群且房屋 较高的城市市区	450	0.30

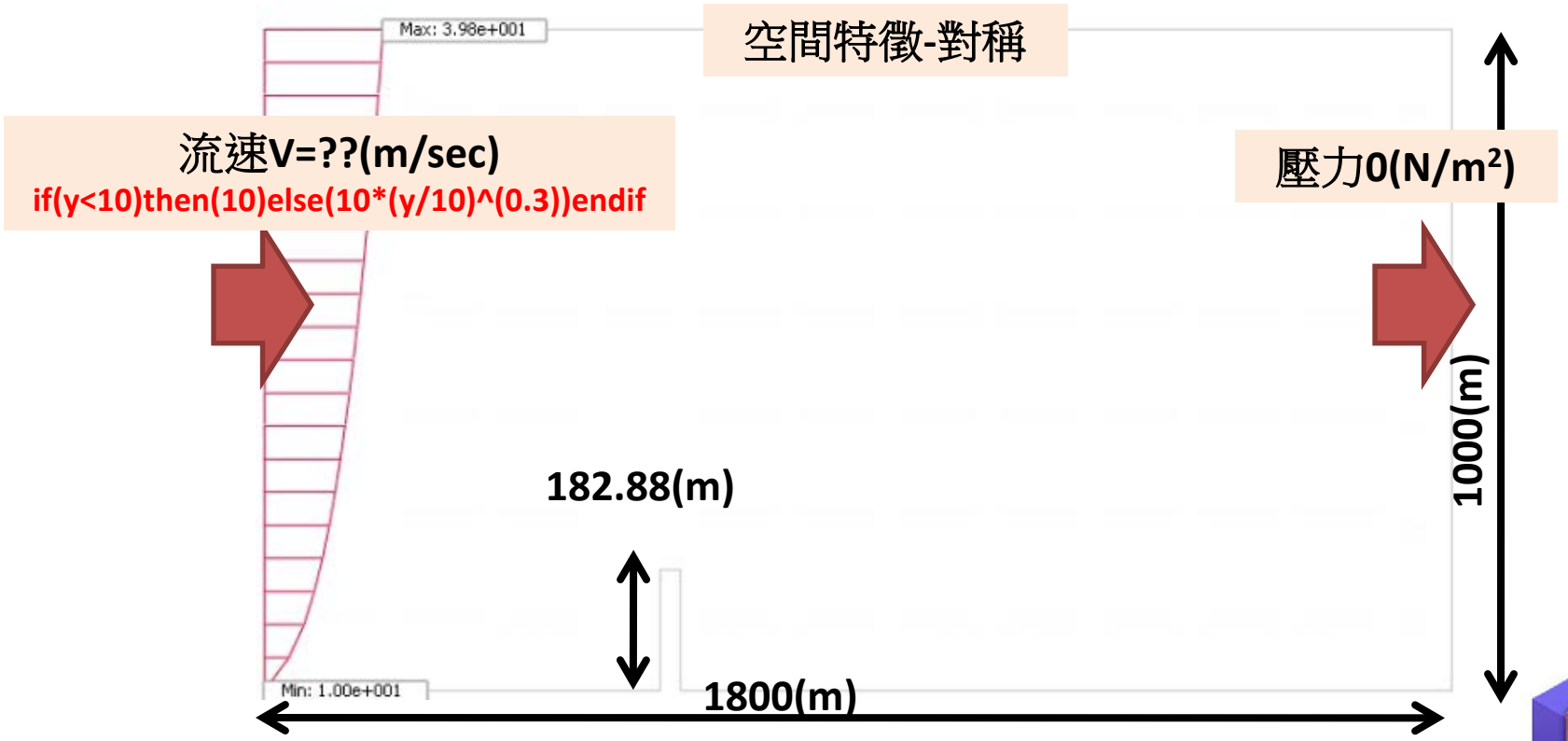
Reference

聶少鋒,周緒紅.CAARC標準高層建築三維鈍體繞流風場數值模擬.土木建築與環境工程,2009,Vol.31 No.6

分析說明

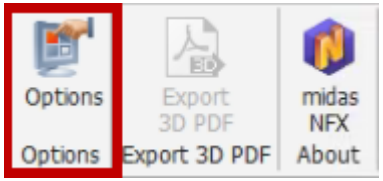
Z_0:參考高度Z_0:10m
U_0:參考高度風速U_0:10(m/s)
地面粗糙度指數α :0.3(市區)

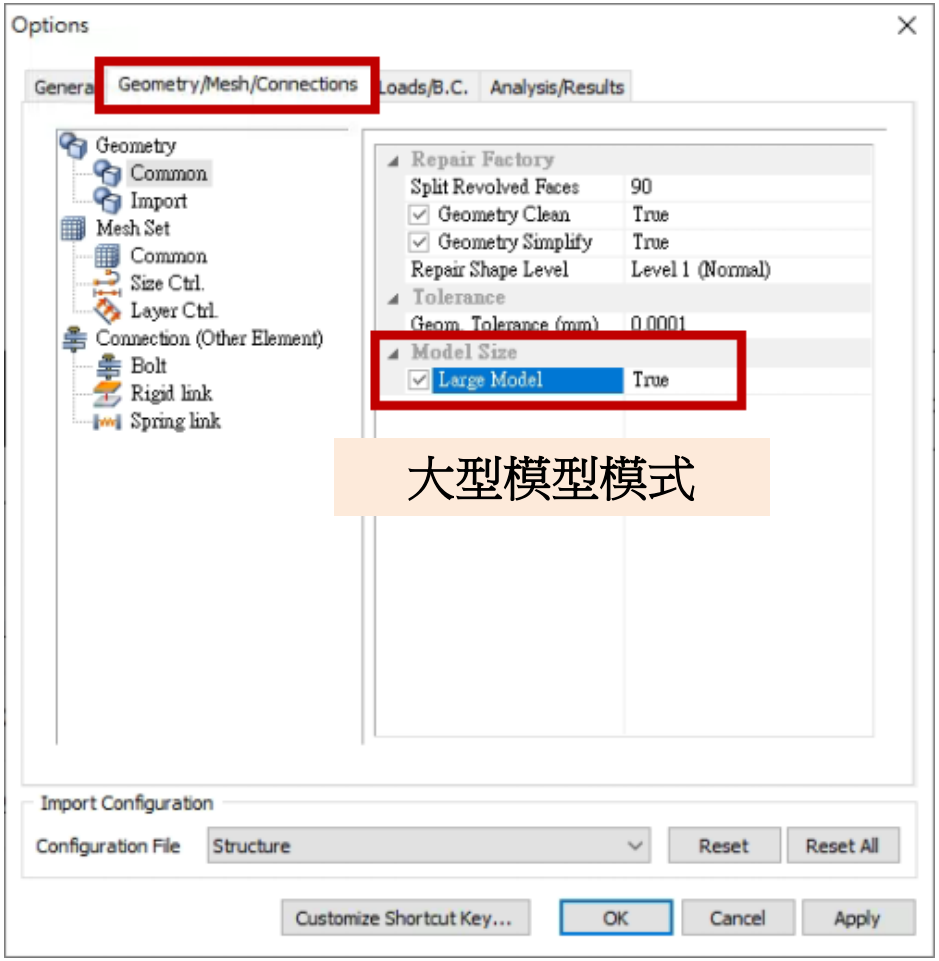
C_p風壓係數 ??
(大樓2/3H高位置)



註:2D空間高度座標為y軸。

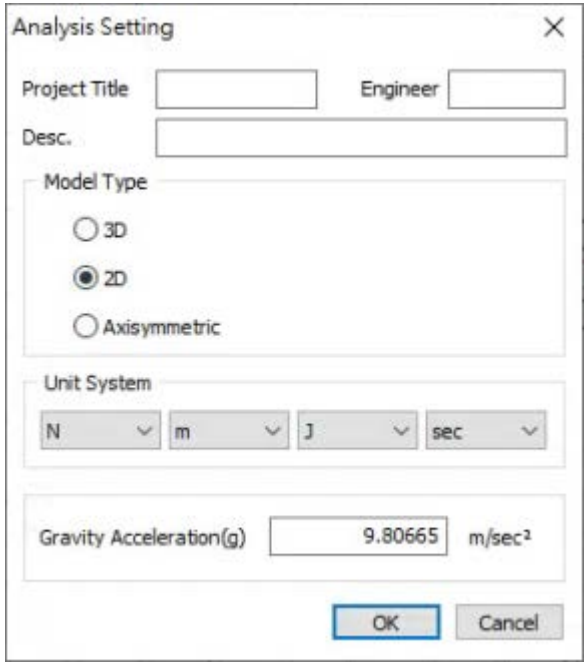
環境-1



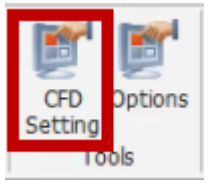


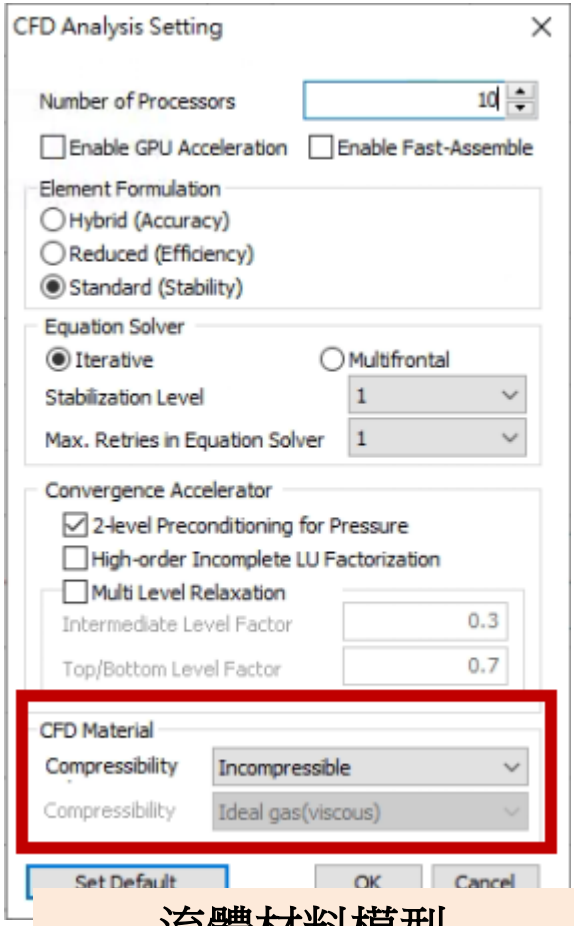
註:標準模型模式1000m ×1000m ×1000m

環境-2



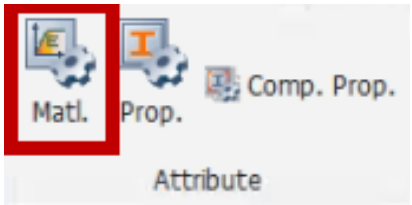
單位使用N/m/J/sec

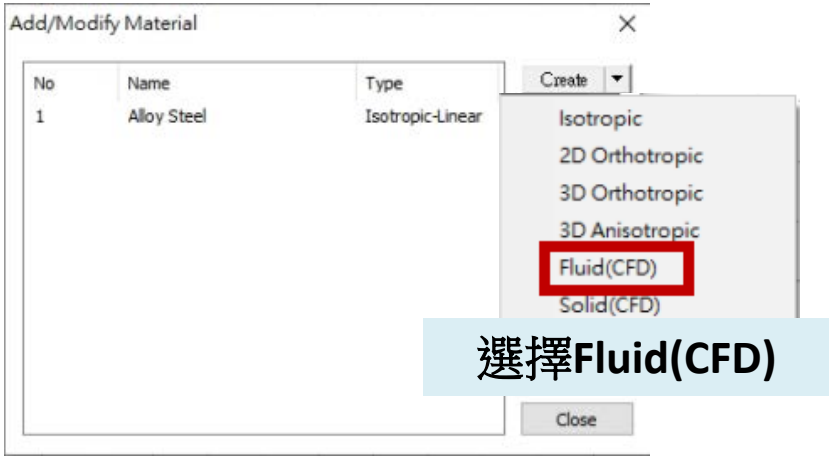




流體材料模型
不可壓縮流

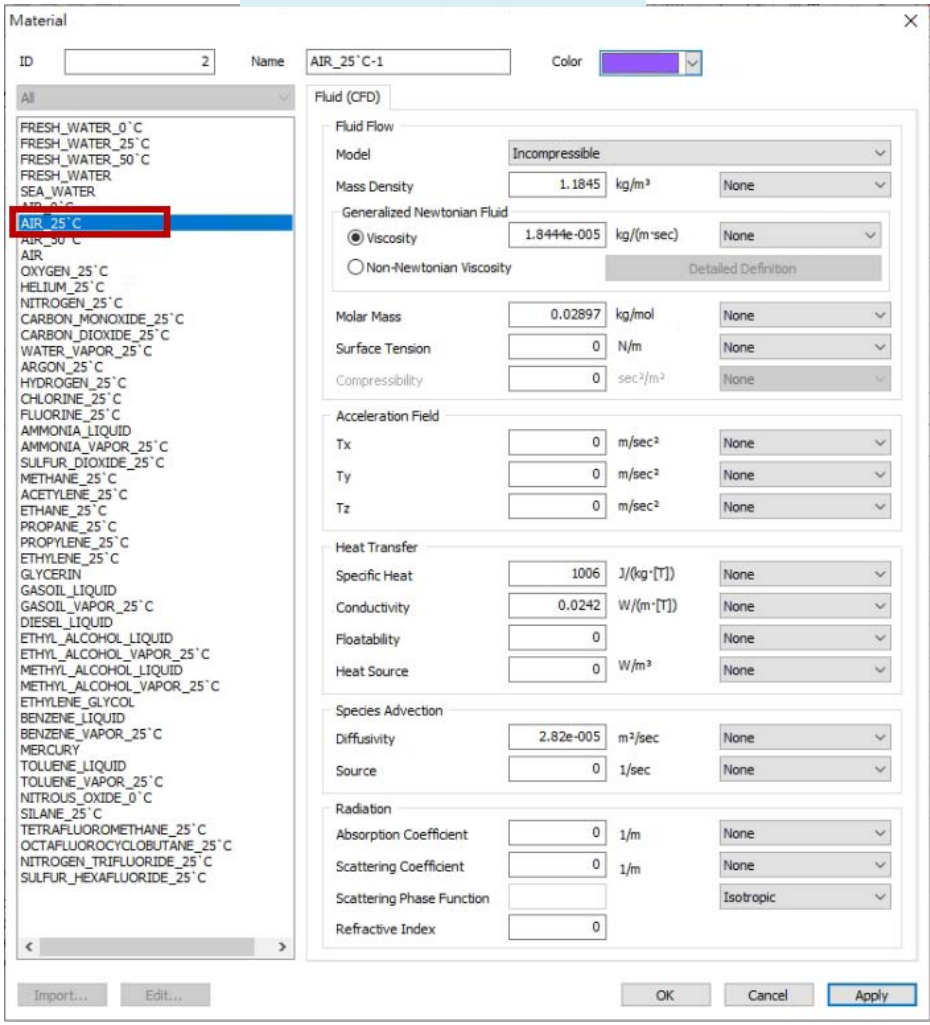
材料





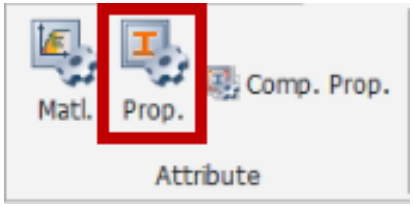
選擇Fluid(CFD)

新增AIR 25 ℃

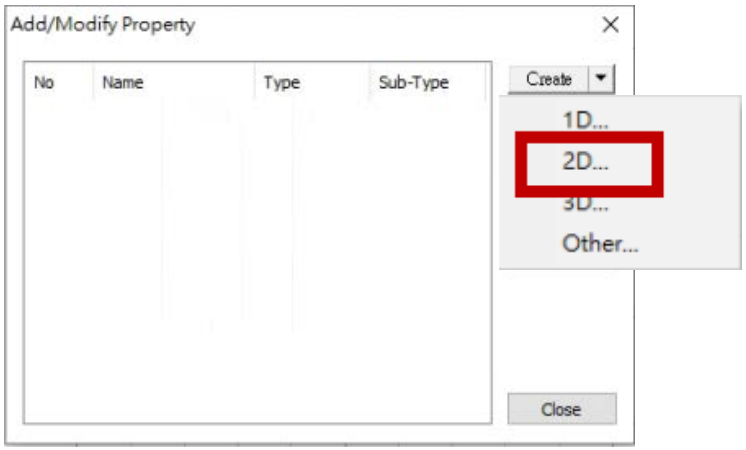


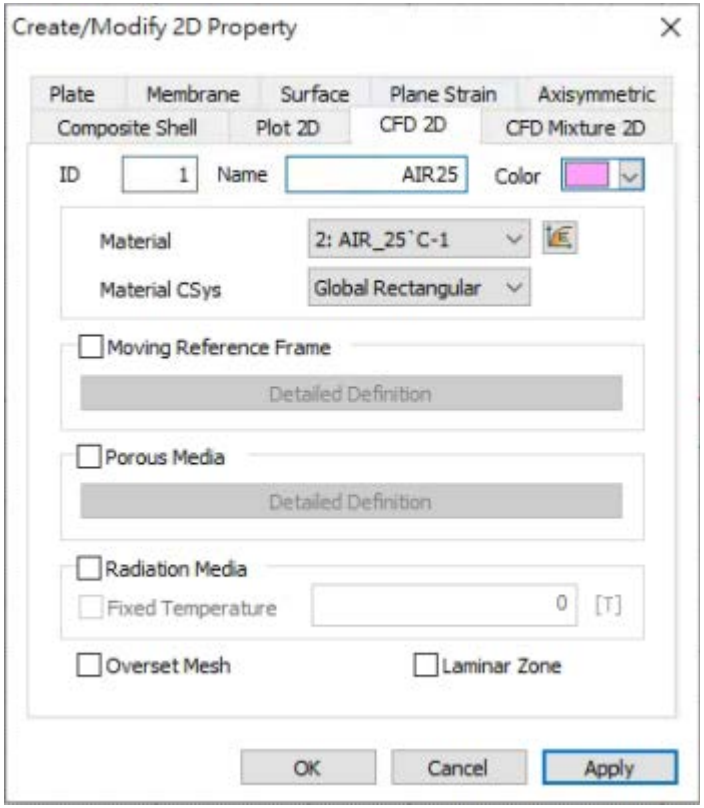
註:NFX CAE/CFD是整合性軟體,材料和屬性包含結構/熱/流場。

屬性

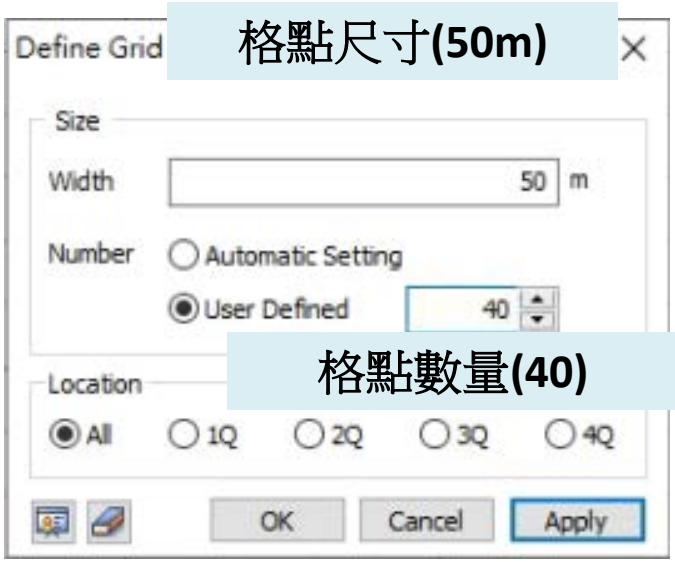


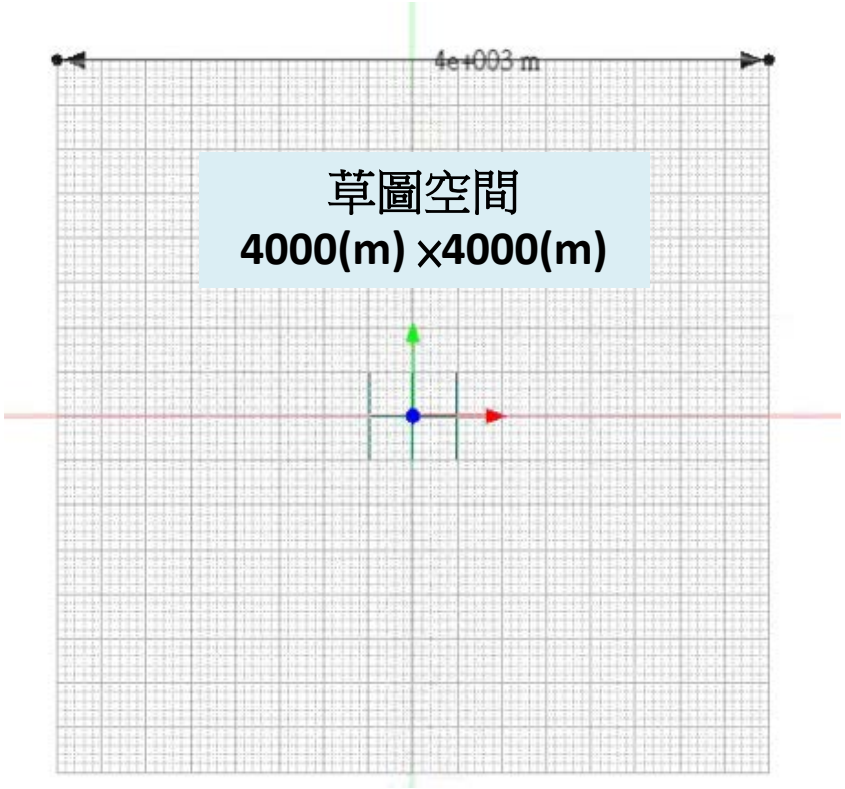
新增AIR 25 ℃ Property





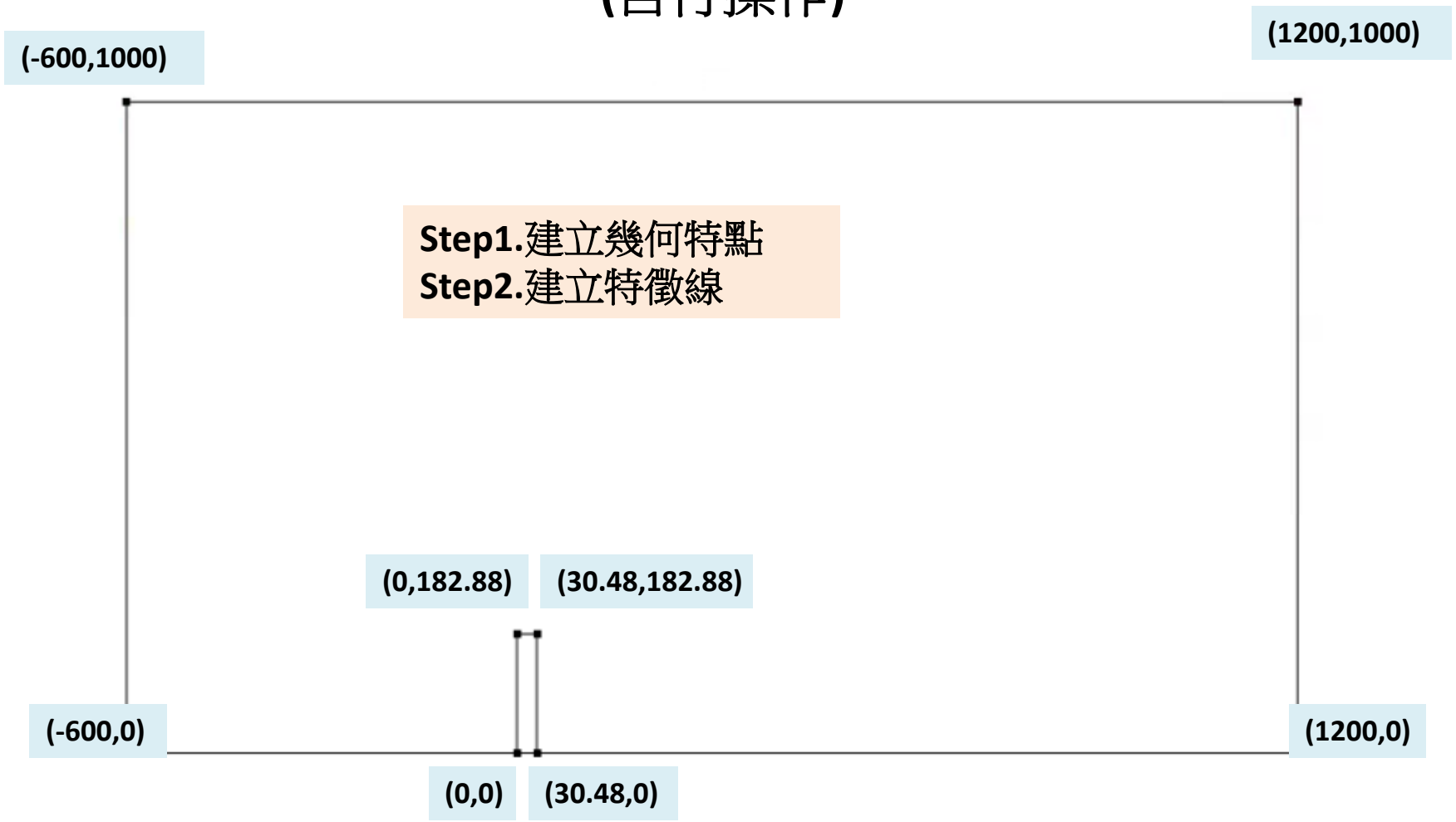
草圖格點


Define Grid
格點尺寸(50m)
Size
Width
50
m
Number
Automatic Setting
User Defined
40
格點數量(40)
Location
All
1Q
2Q
3Q
4Q
OK
Cancel
Apply

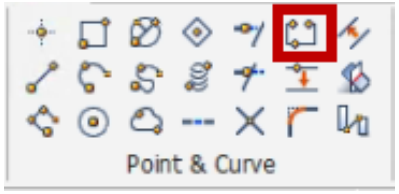

4e+003 m
草圖空間
4000(m) x4000(m)

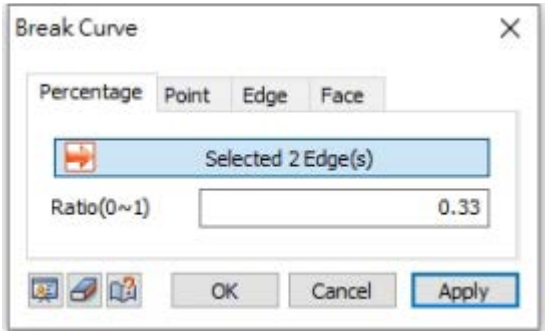
特徵線繪製

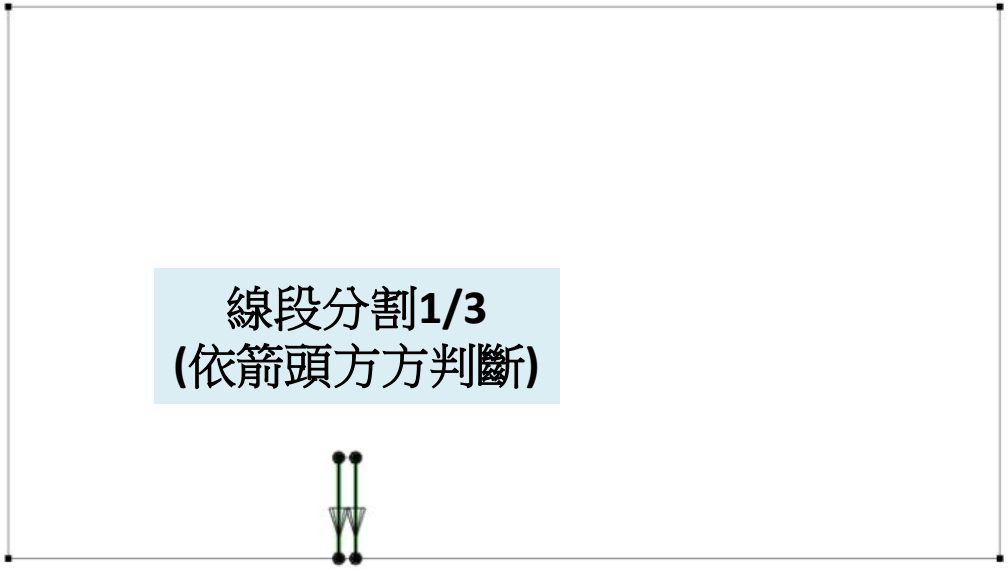
(自行操作)


(-600,1000)
(1200,1000)
Step1.建立幾何特點
Step2.建立特徵線
(0,182.88)
(30.48,182.88)
(-600,0)
(1200,0)
(0,0)
(30.48,0)

特徵線分割

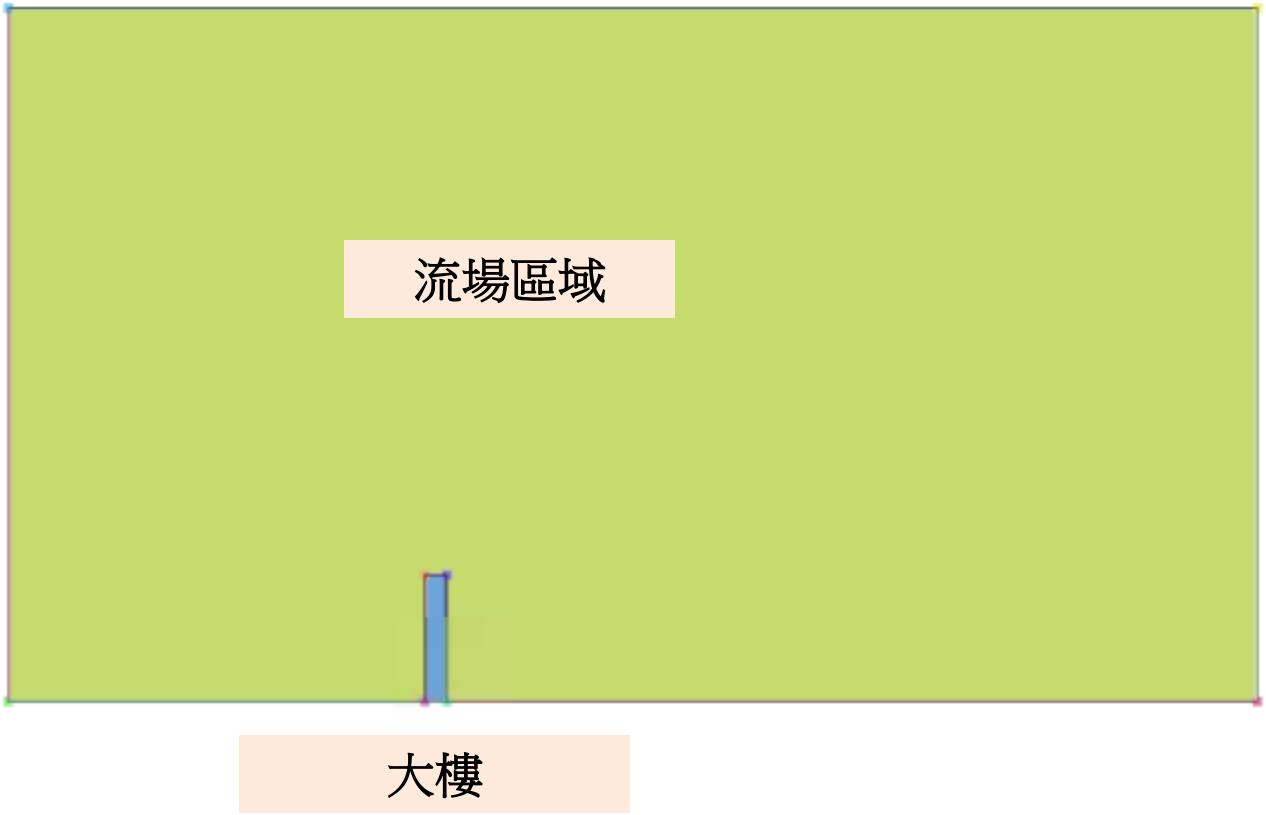

Point & Curve


Break Curve
Percentage
Point
Edge
Face
Selected 2 Edge(s)
Ratio(0~1)
0.33
OK
Cancel
Apply


線段分割1/3
(依箭頭方方判斷)

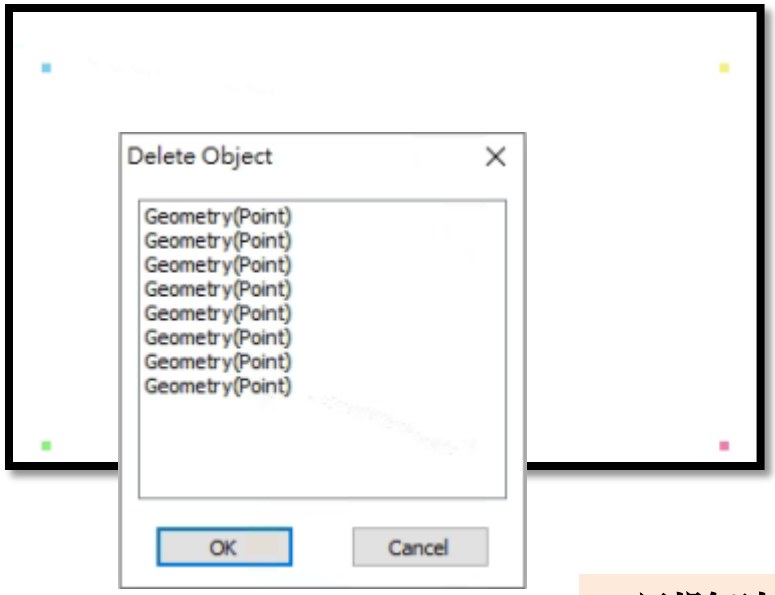
特徵面繪製

(自行操作)

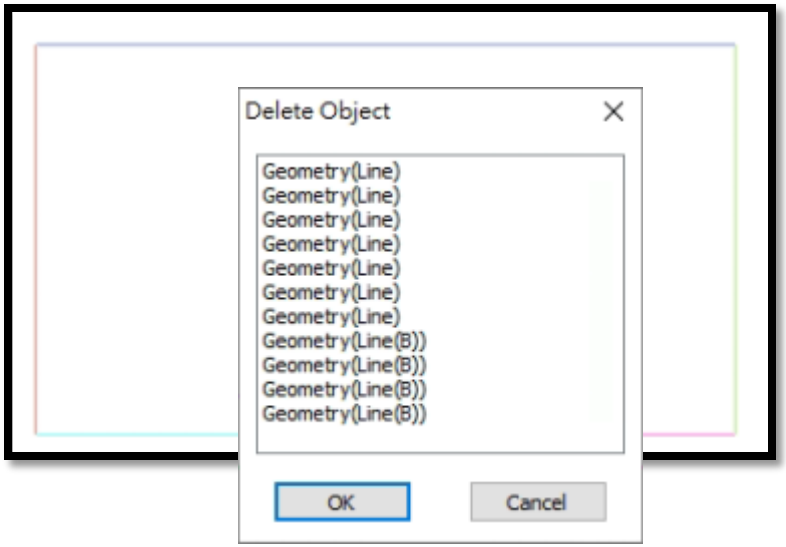


刪除點/線/面特徵

刪除所有點特徵

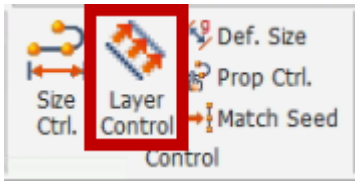


刪除所有線特徵



刪除建物面特徵

邊界層指定



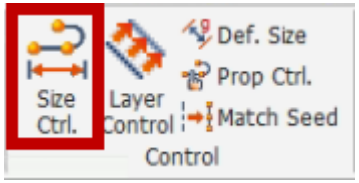
Number of Layers:3
Total Layer Height:0.1(m)
Layer Growth Rate:1

Step1.選取面特徵

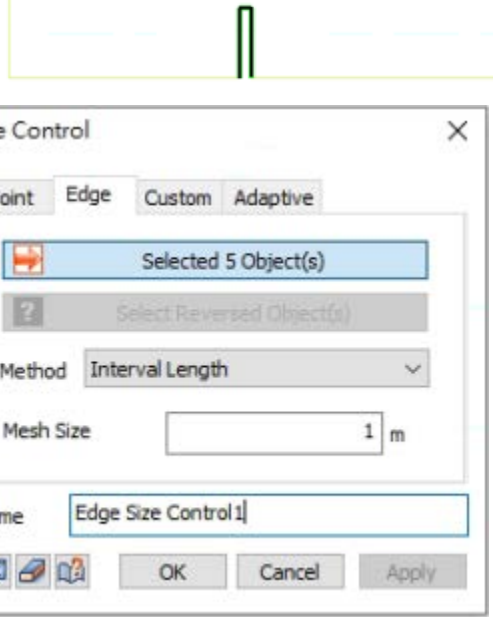
Step2.選取建物和地表線特徵

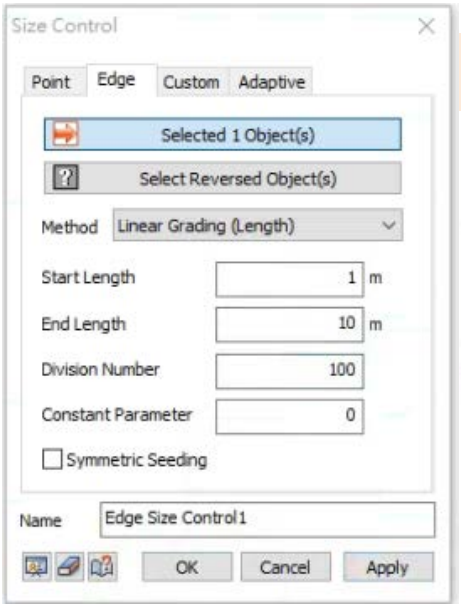
midas NFX-CFD

網格尺寸指定

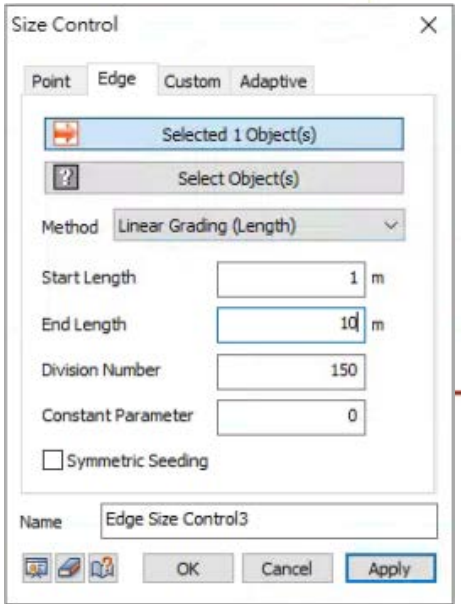


選取建物線特徵





選取地表左側線特徵



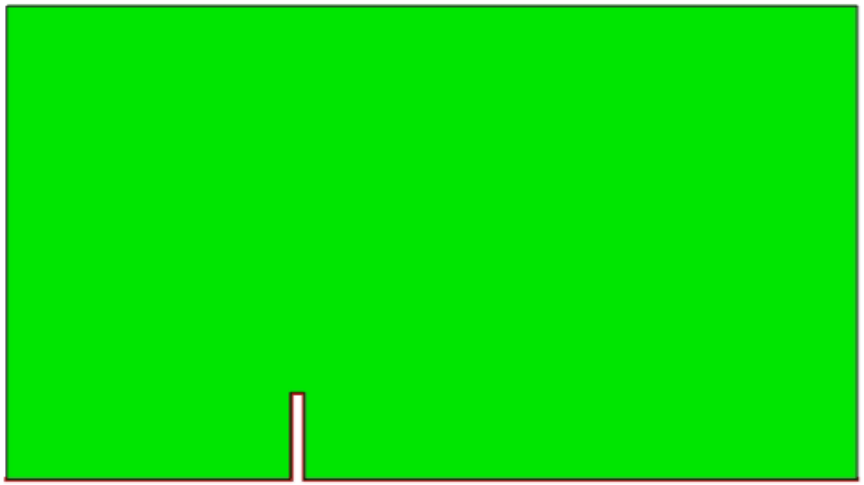
選取地表右側線特徵

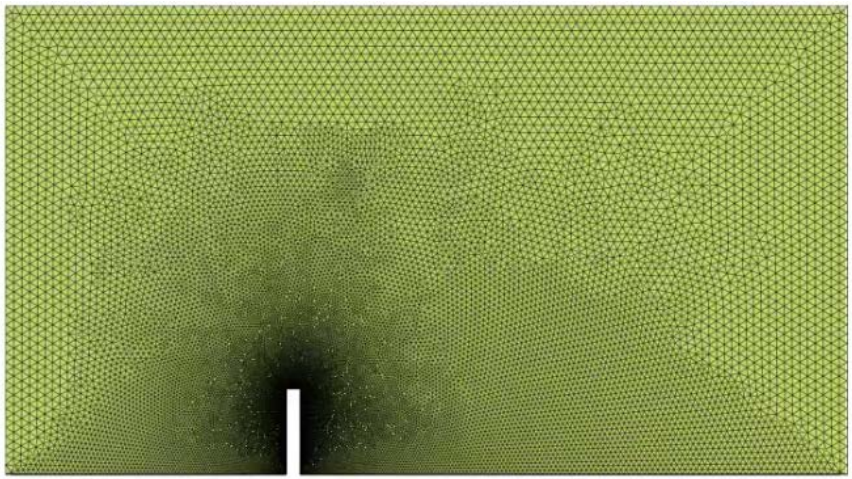
MIDAS

midas NFX-CFD

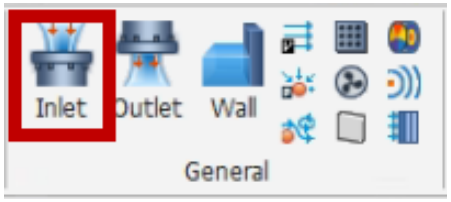
2D網格

三邊形網格/網格尺寸20(m)



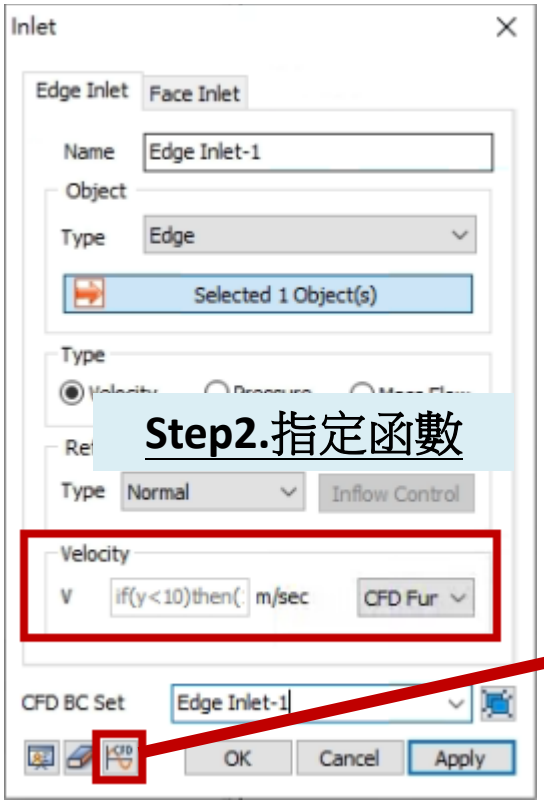


入流



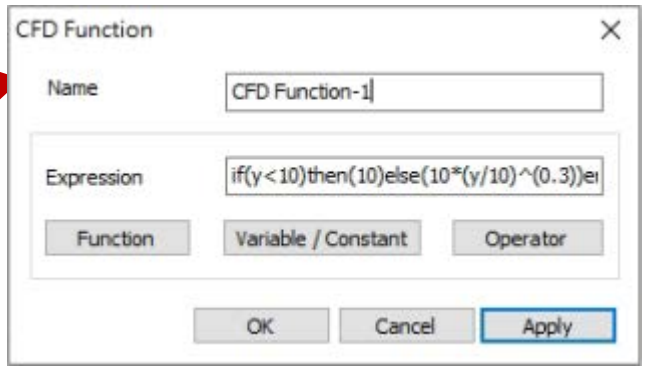
入流流速-函數定義

if(y<(10))then(10)else(10*((y)/10)^(0.3))endif



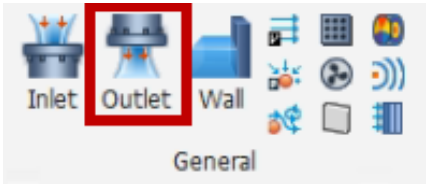
Step2.指定函數

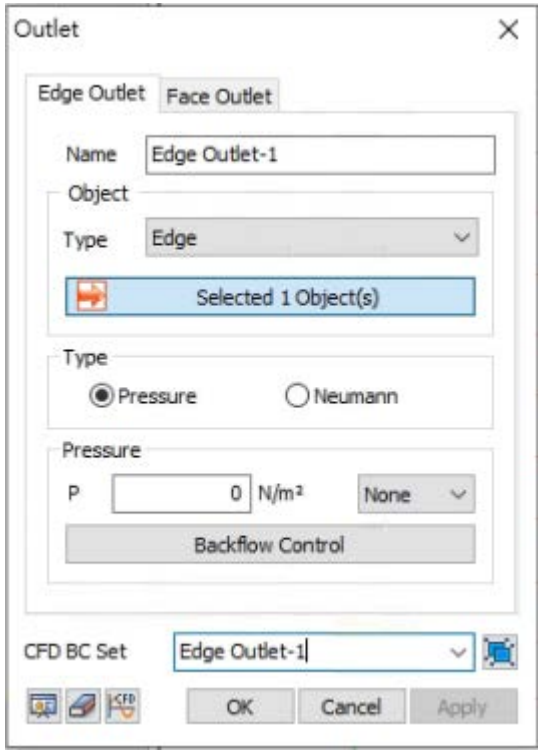
Step1.新增風廓線入流函數



註:不同邊界&載荷建議使用不同集合名稱。

出流

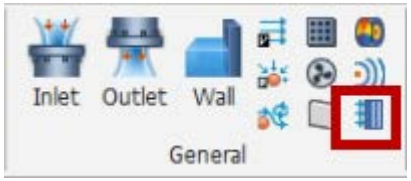


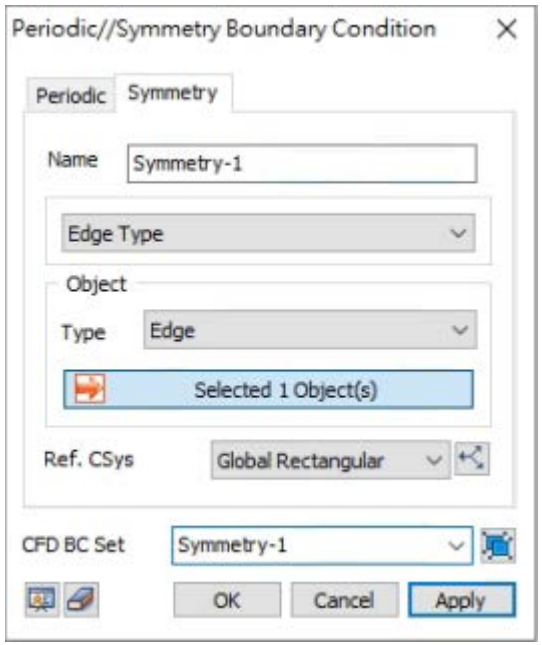


壓力0(N/m²)

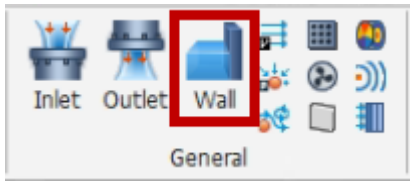
註:不同邊界&載荷建議使用不同集合名稱。

對稱

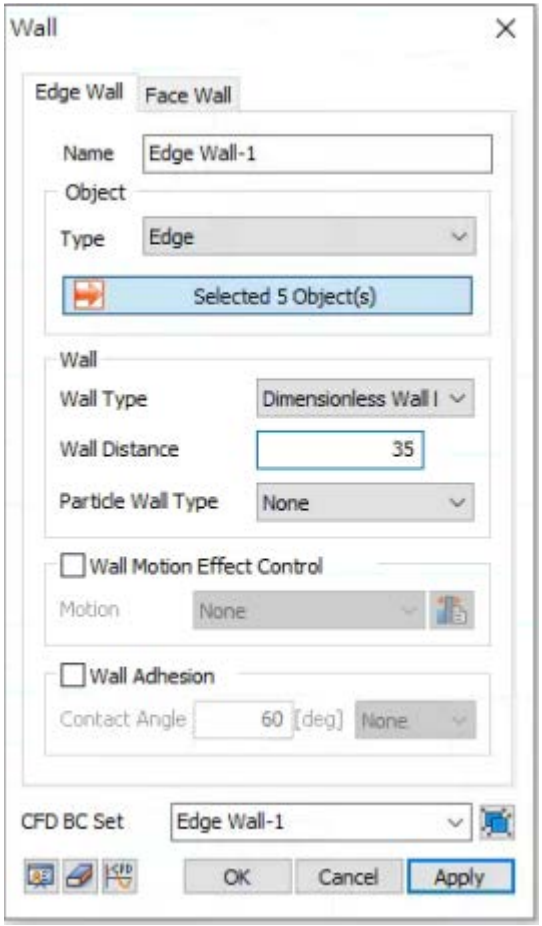




註:不同邊界&載荷建議使用不同集合名稱。

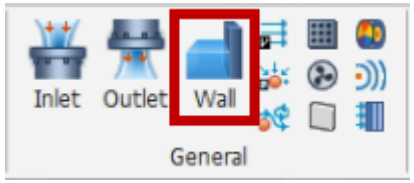


壁函數-1
(Dimensionless Wall Applied)

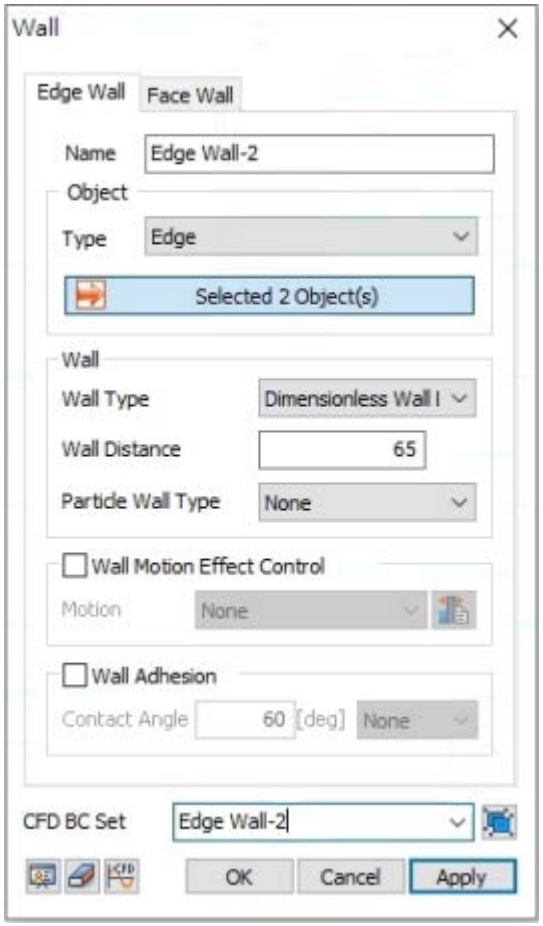


選取建物線特徵

壁函數(Dimensionless Wall Applied)
Y+=35

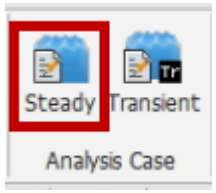


壁函數-2
(Dimensionless Wall Applied)



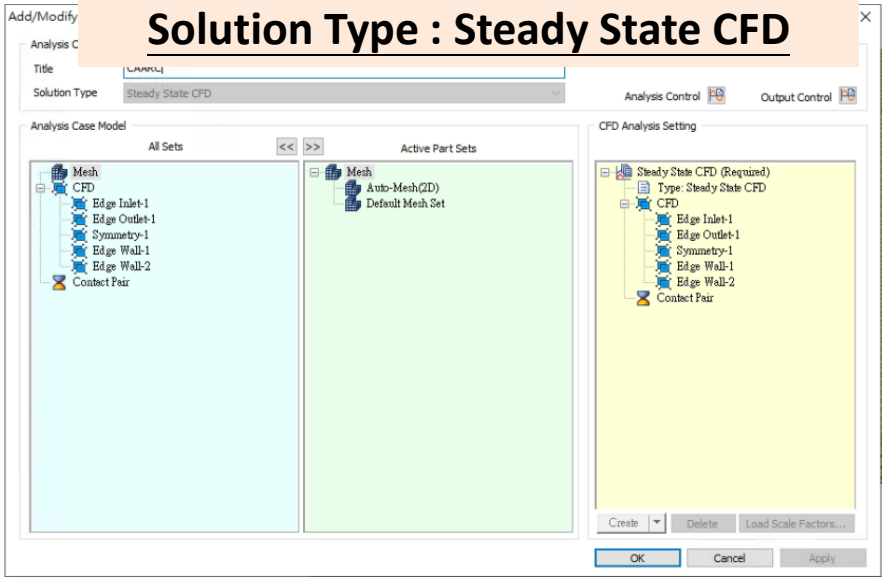
選取地表線特徵

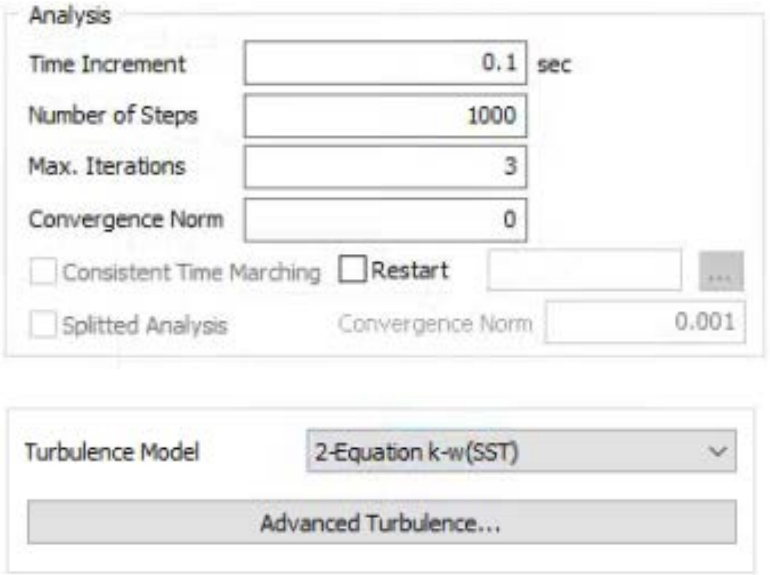
壁函數(Dimensionless Wall Applied)
Y+=65



CFD穩態分析
(CAARC建築風場數值模擬)

Solution Type : Steady State CFD





分析結果
(最後1增量)

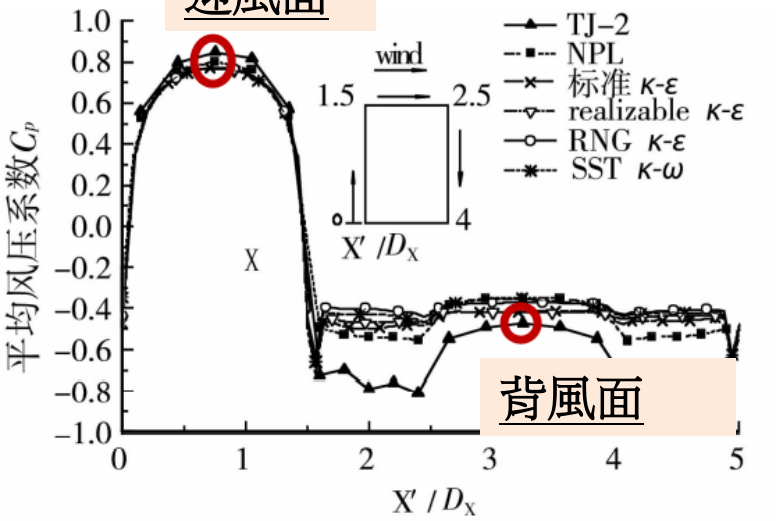
參考
聶少鋒,周緒紅.CAARC標準高層建築三維鈍體繞流風場數值模擬.

$$C_P = 2(P - P_0)/\rho U_0^2$$

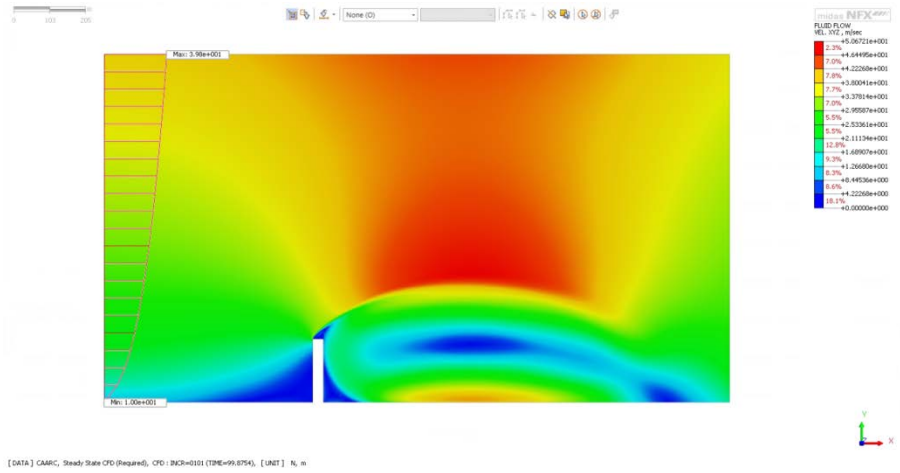
2D迎風面
P:295.7097(N/m²)
P_0:不考慮
ρ : 1.1845(kg/m³)
U_0:24(m/sec)
H:182.88(m)
C_p:0.8668

2D背風面
P:-309.5183(N/m²)
P_0:不考慮
ρ : 1.1845(kg/m³)
U_0:24(m/sec)
H:182.88(m)
C_p:-0.9058

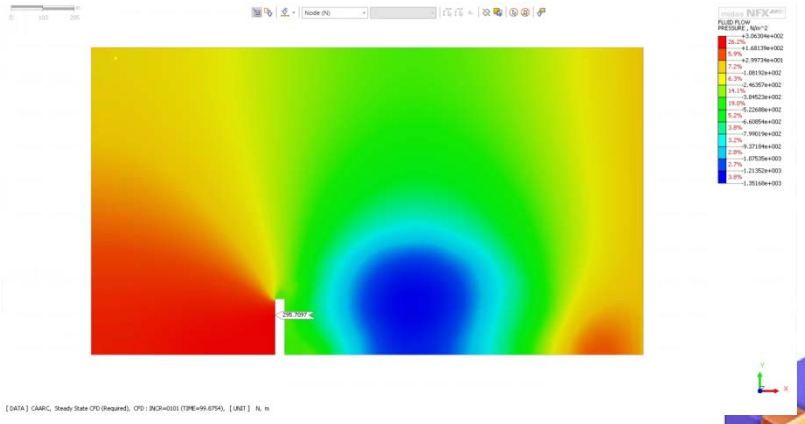
迎風面



流速(m/sec)



壓力(N/m²)



註:2維空間分析流場未考慮側邊流動，風壓係數誤差會偏大。

補充3D CAARC建築風場

(第800增量)

3D迎風面

P:254.1(N/m²)

C_p:0.7449

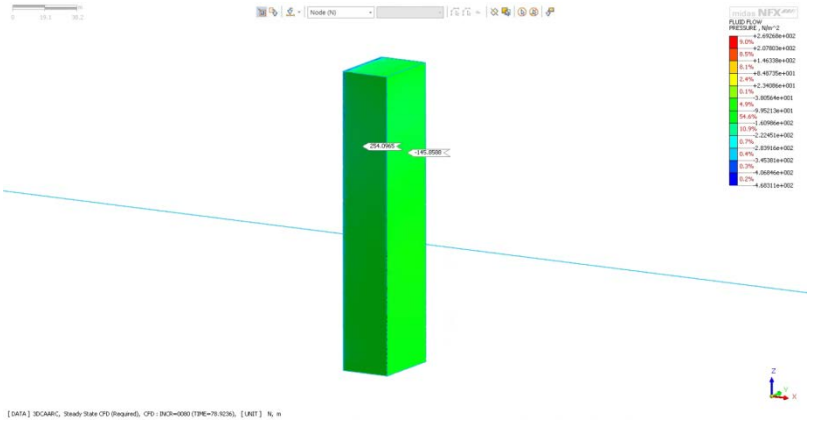
3D背風面

P:-145.9N/m²)

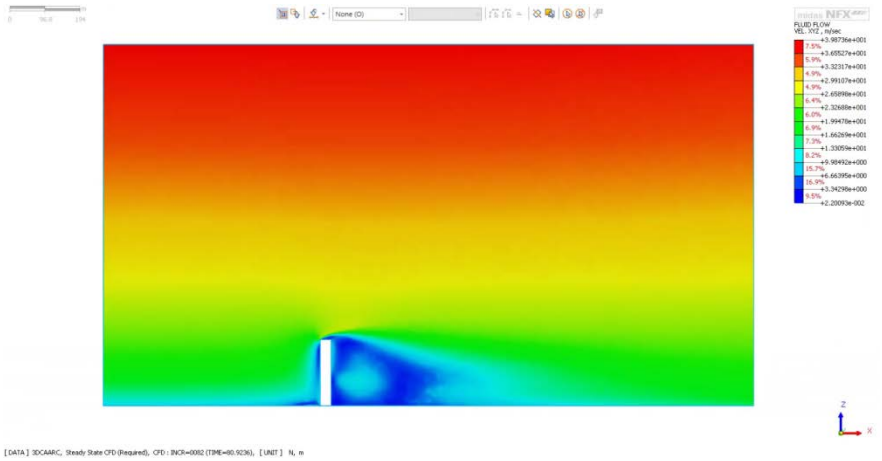
C_p:-0.4277

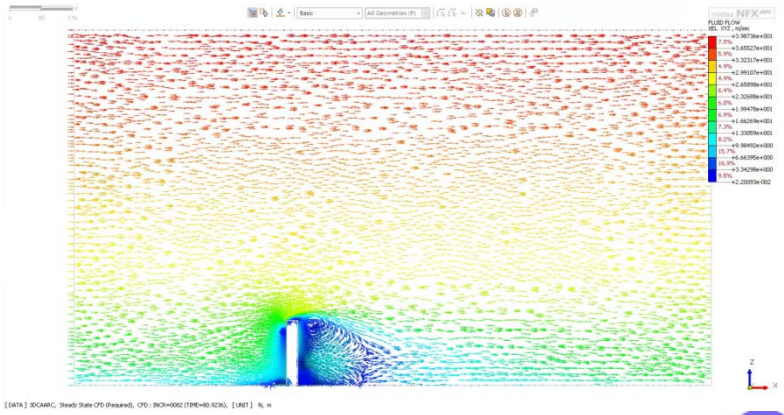
接近文獻結果

壓力(N/m²)



流速(m/sec)





註:3維空間分析計算所需時較長。

CFD_標準教學系列

強制對流分析101-3D內流場分析

台灣邁達斯

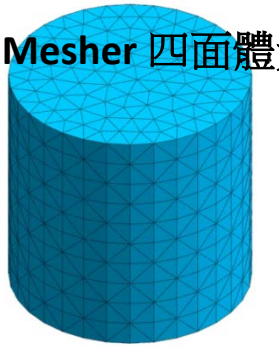
NFX CFD 3D元素類型

三維元素

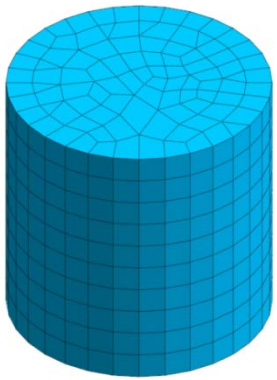
可以考慮具有 4、5、6 或 8 個節點的四面體、五面體或六面體元素，其中五面體元素可以定義為楔形元素或金字塔形元素。

3D網格劃分方式

Tetra Mesher 四面體元素



Hybrid Mesher(混合網格)=>六面體元素為主+五面體元素



NFX CFD不支援高階(2階)元素

midas NFX-CFD

分析說明

Wall Function(外側壁面)

(Dimensionless Wall Distance y+=65)

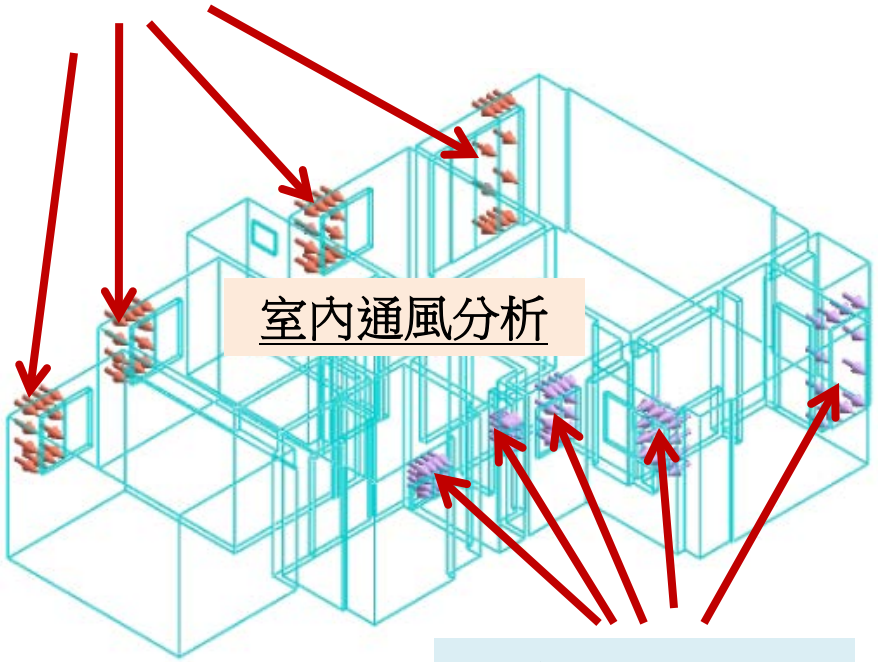
空氣_AIR_25 ℃

Fluid Flow

Model	Incompressible		
Mass Density	1.1845	kg/m³	None
Generalized Newtonian Fluid			
Viscosity	1.8444e-005	kg/(m·sec)	None
Non-Newtonian Viscosity	Detailed Definition		

入流口V=1m/sec

流域模型建立

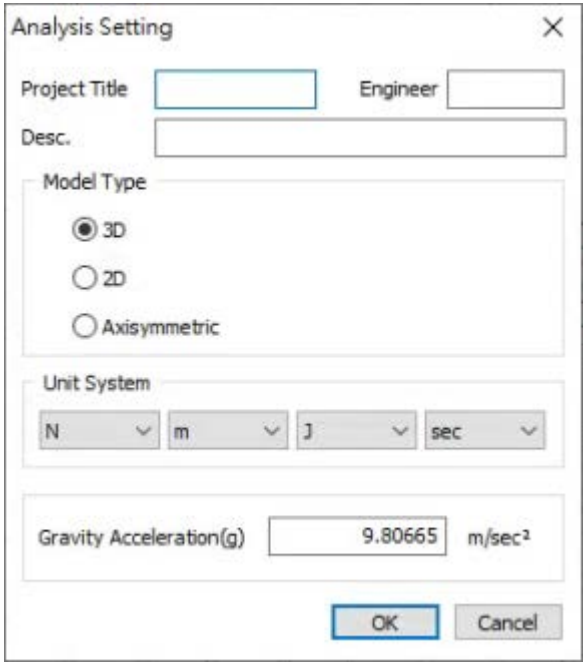


室內通風分析

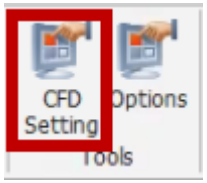
出流口P=0(N/m²)

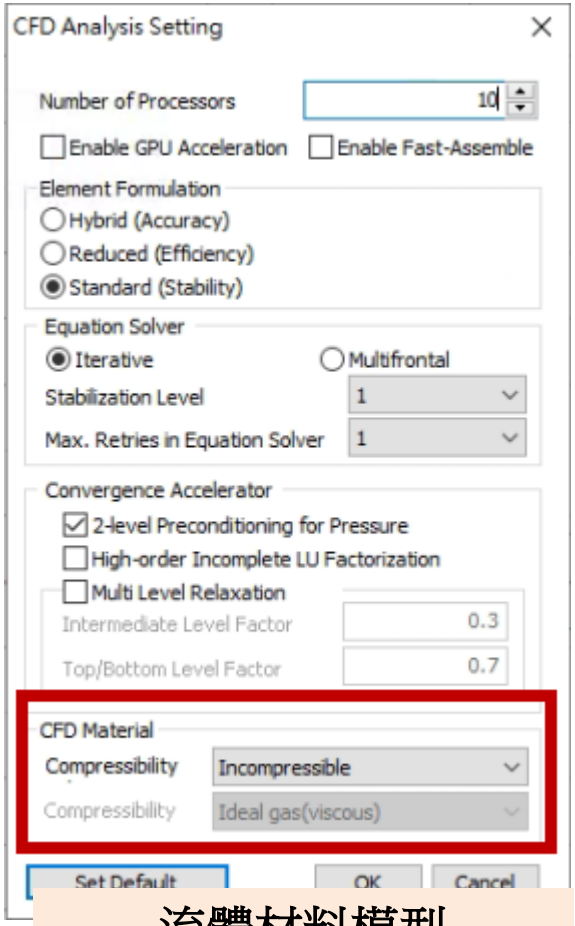
MIDAS

環境



單位使用N/m/J/sec

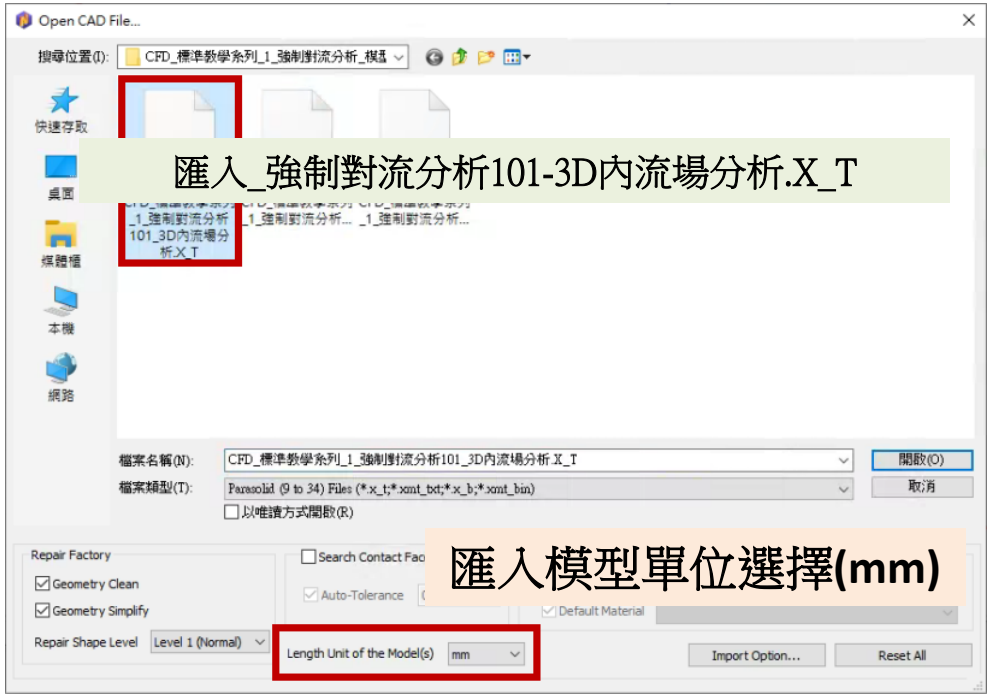


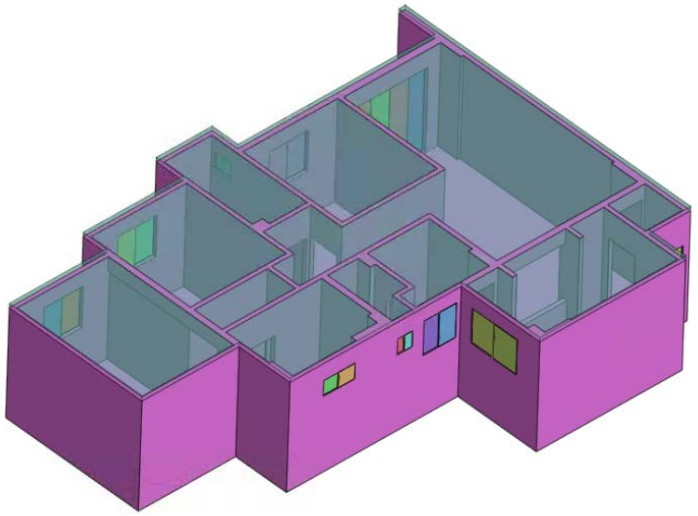


流體材料模型
不可壓縮流

匯入模型


CAD
CAD
Import
Export
CAD File

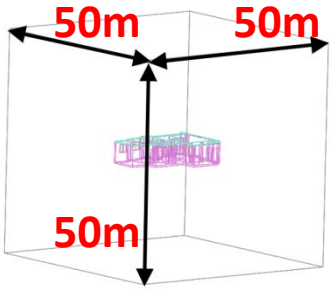

Open CAD File...
搜尋位置(I):
CFD_標準數學系列_1_強制對流分析_模型
快速存取
桌面
媒體櫃
本機
網路
匯入_強制對流分析101-3D內流場分析.X_T
檔案名稱(N):
CFD_標準數學系列_1_強制對流分析101_3D內流場分析.X_T
檔案類型(T):
Parasolid (9 to 34) Files (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
開啟(O)
取消
以唯讀方式開啟(R)
匯入模型單位選擇(mm)
Repair Factory
Geometry Clean
Geometry Simplify
Repair Shape Level
Level 1 (Normal)
Search Contact Fac
Auto-Tolerance
Default Material
Length Unit of the Model(s)
mm
Import Option...
Reset All

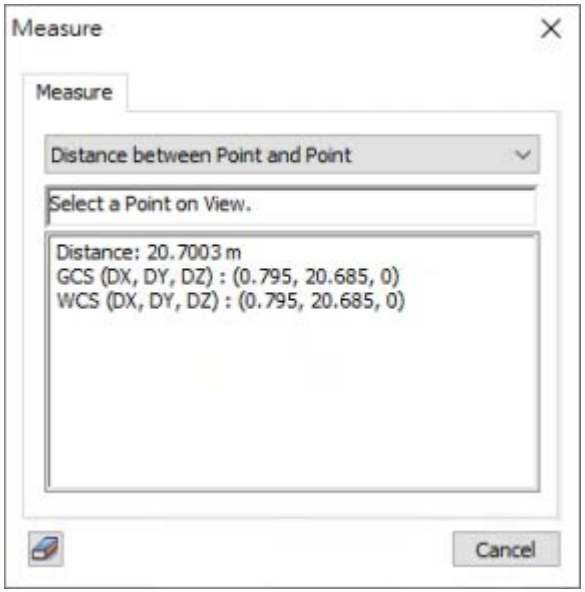


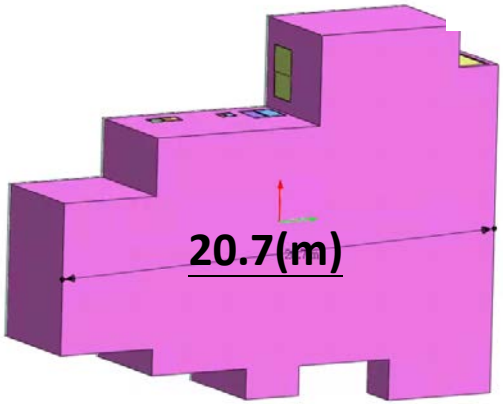
模型尺寸量測

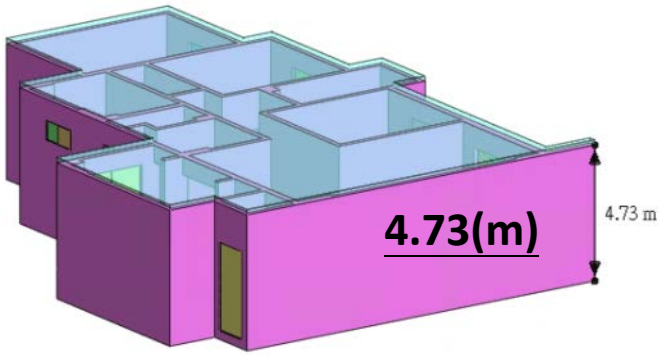
估算建立流場區域空間

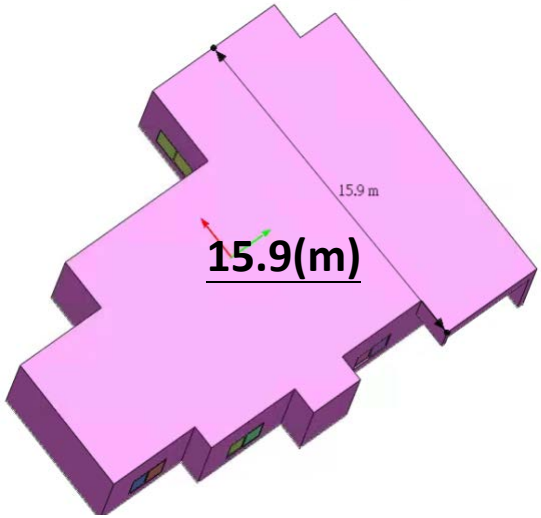
50m×50m×50m





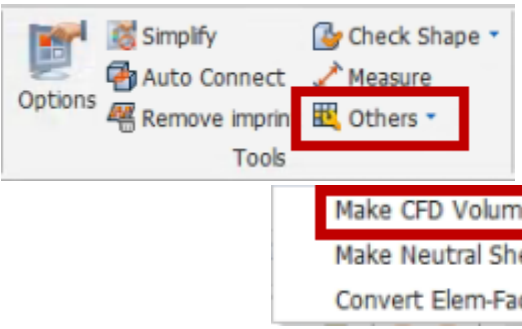




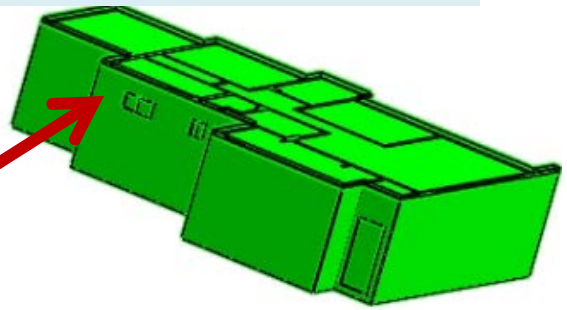


midas NFX-CFD

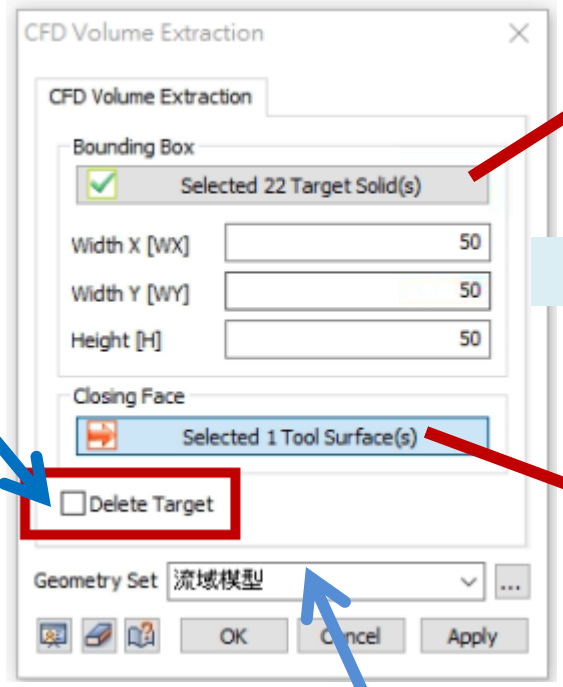
流域模型建立



流域內之特徵
選取全部實體特徵

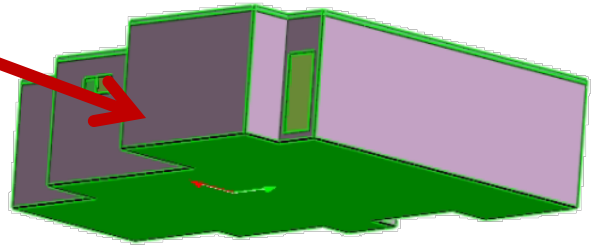


流域內原有幾何是否刪除
(建議保留)



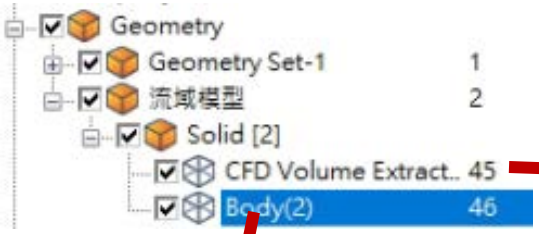
流場區域空間 50m×50m×50m

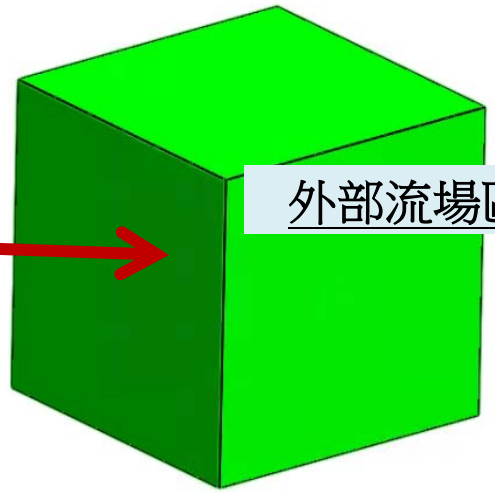
底部面作為參考特徵基準



生成模型放置流域模型幾何集

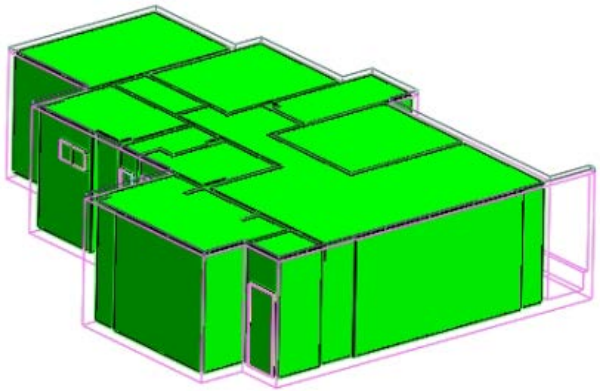
幾何刪除





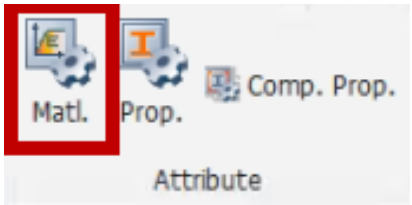
外部流場區域=>刪除

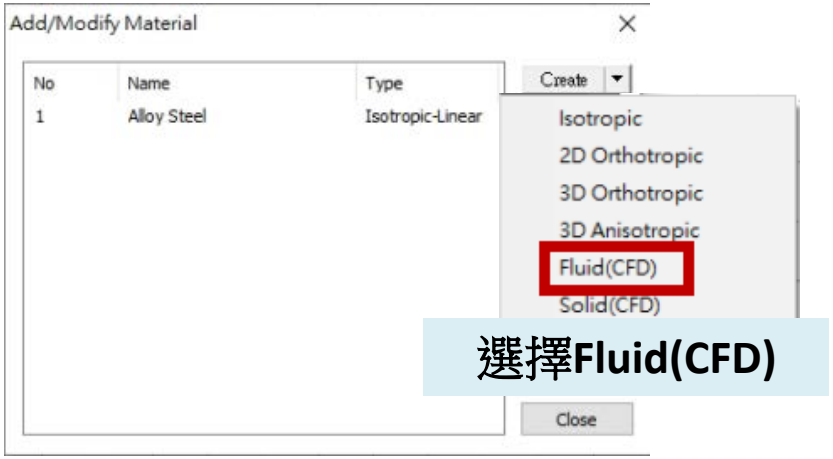
室內流場區域=>保留



手動變更流域幾何顏色顯示
(預設灰色)

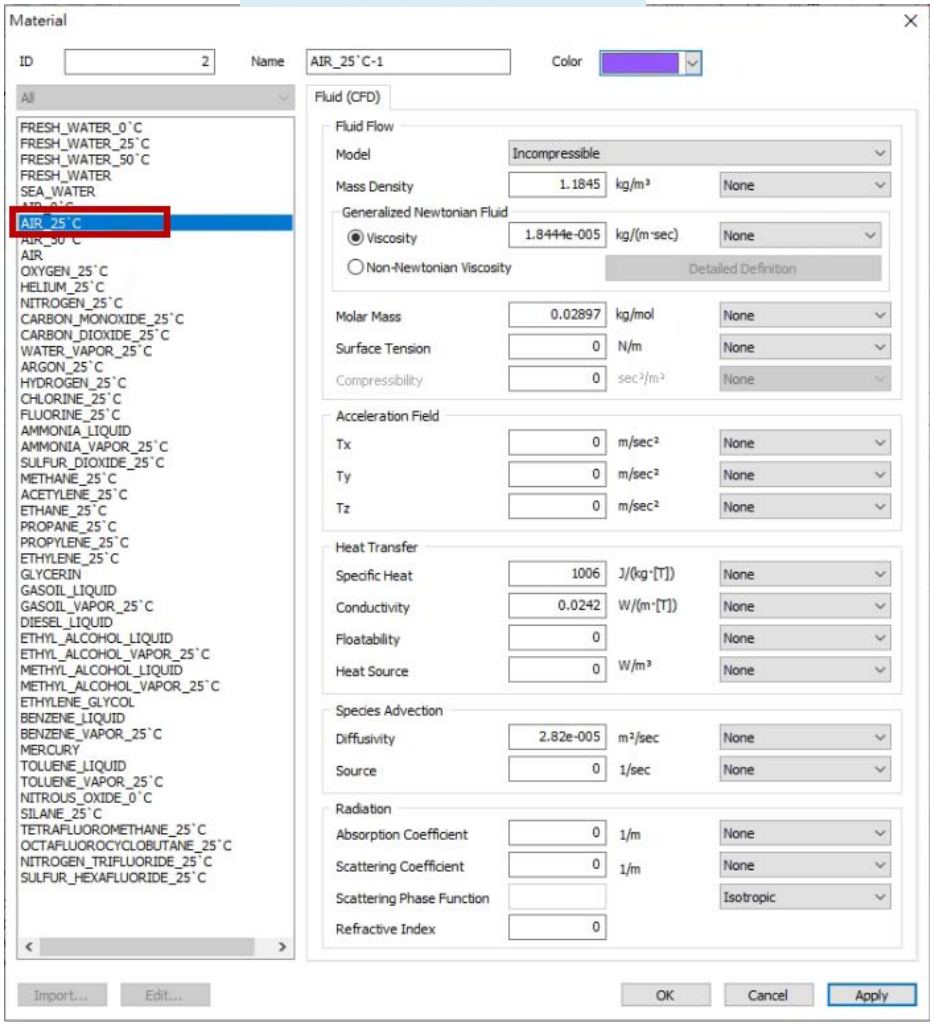
材料





選擇Fluid(CFD)

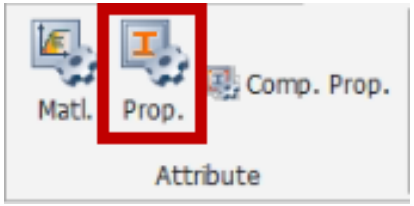
新增AIR 25 ℃

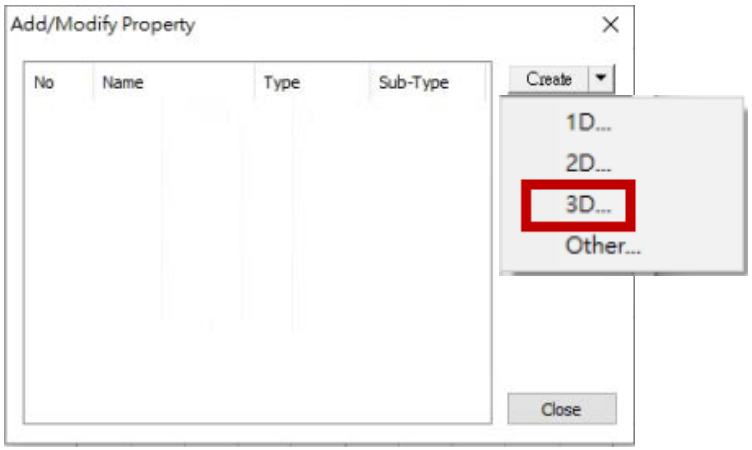


註1:CFD流體使用Fluid(CFD),固體使用Solid(CFD)

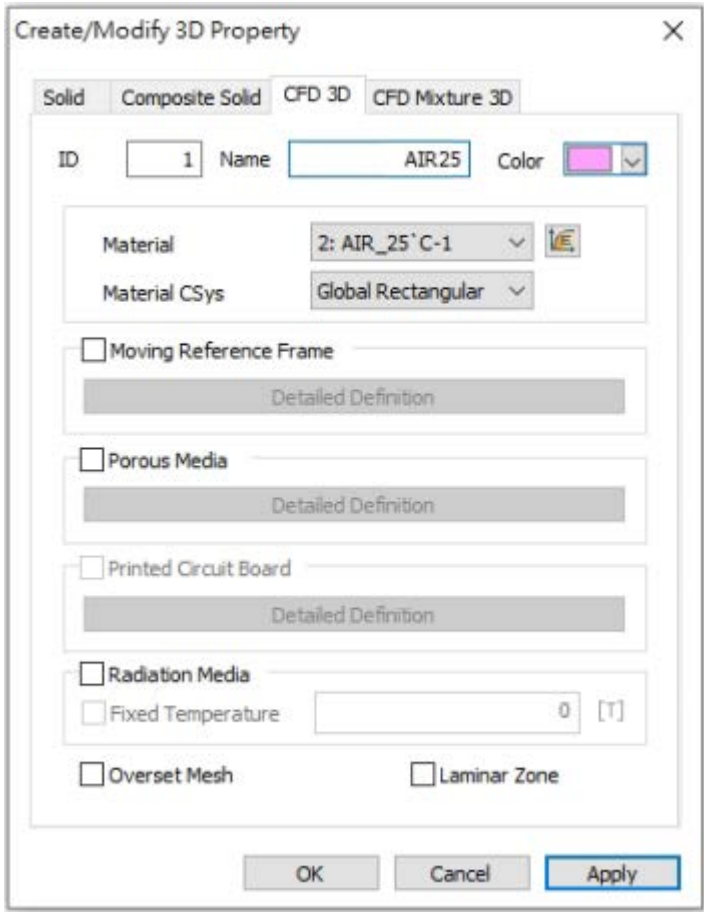
註2:大部份建築風場問題可直接將空氣視為不可壓縮流計算

屬性



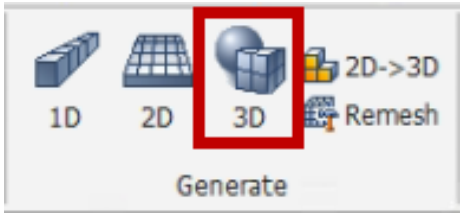


新增AIR 25 ℃ Property

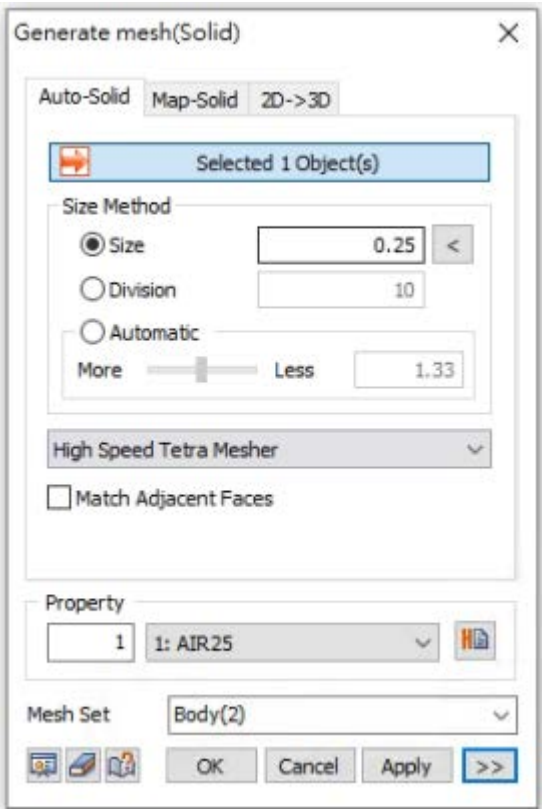


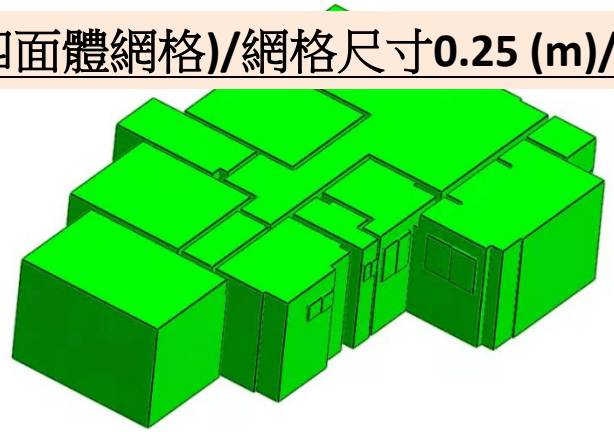
midas NFX-CFD

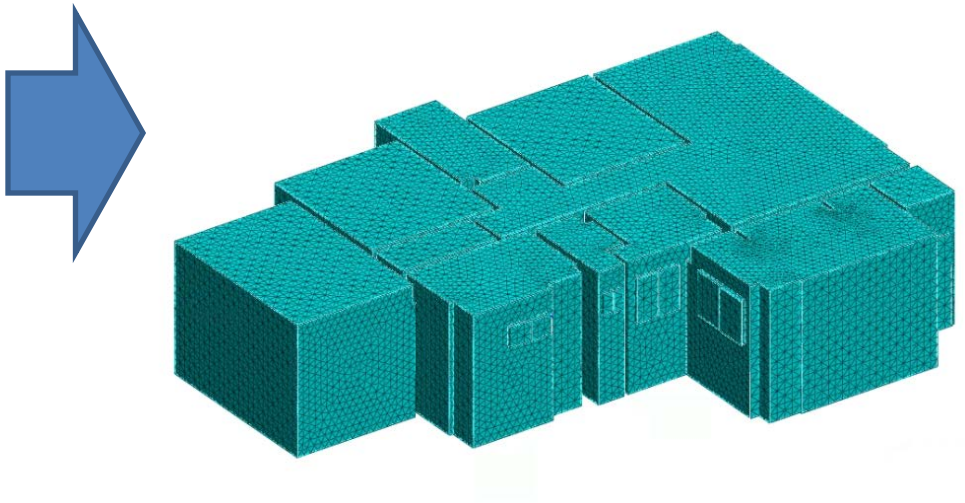
網格



Tetra Mesher(四面體網格)/網格尺寸0.25 (m)/1階元素



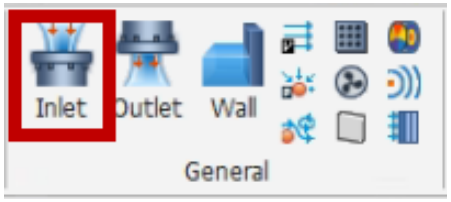


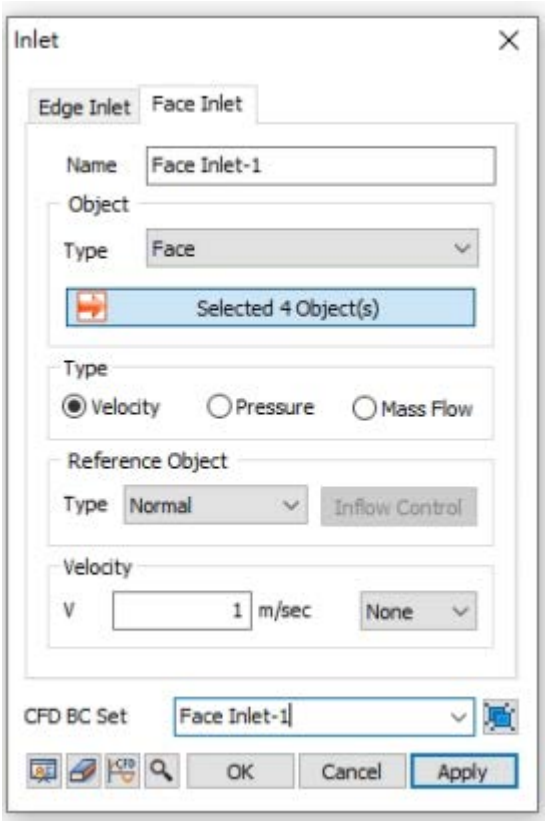


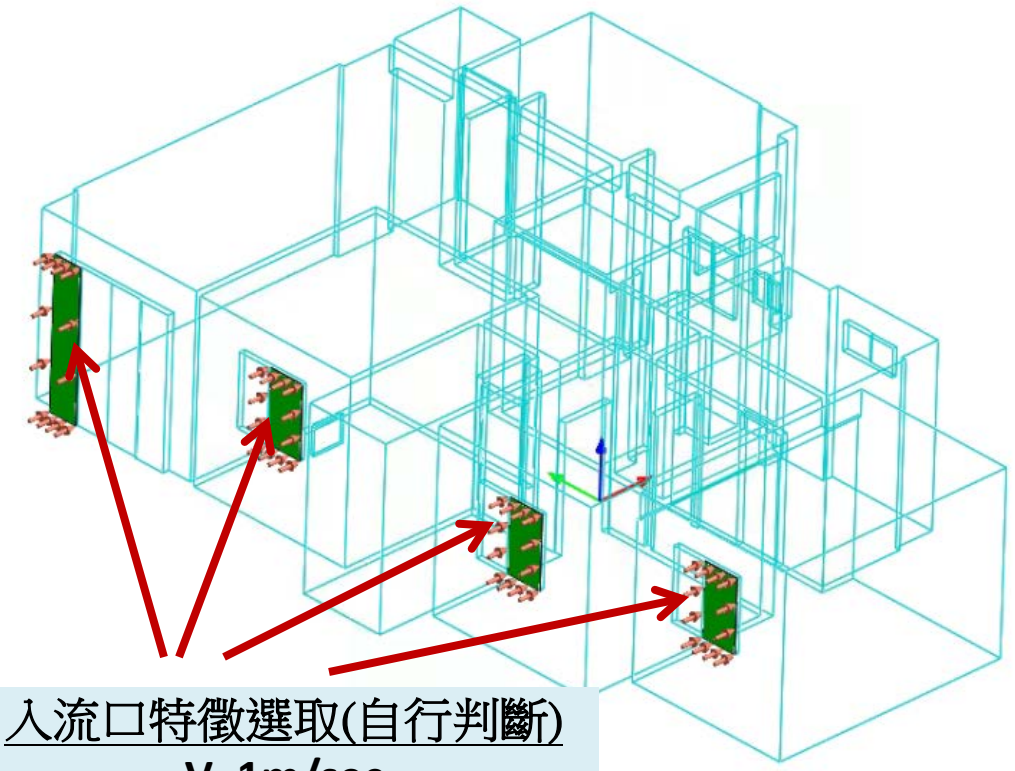
NFX CFD不支援二階(高階)元素

MIDAS

入流





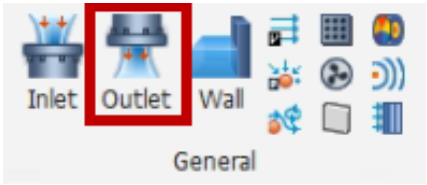


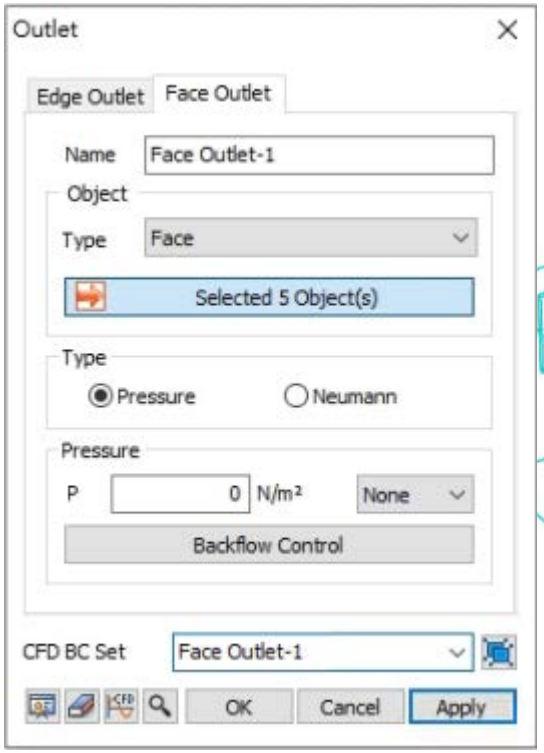
入流口特徵選取(自行判斷)

V=1m/sec

註:Inlet流速預設採Normal方向

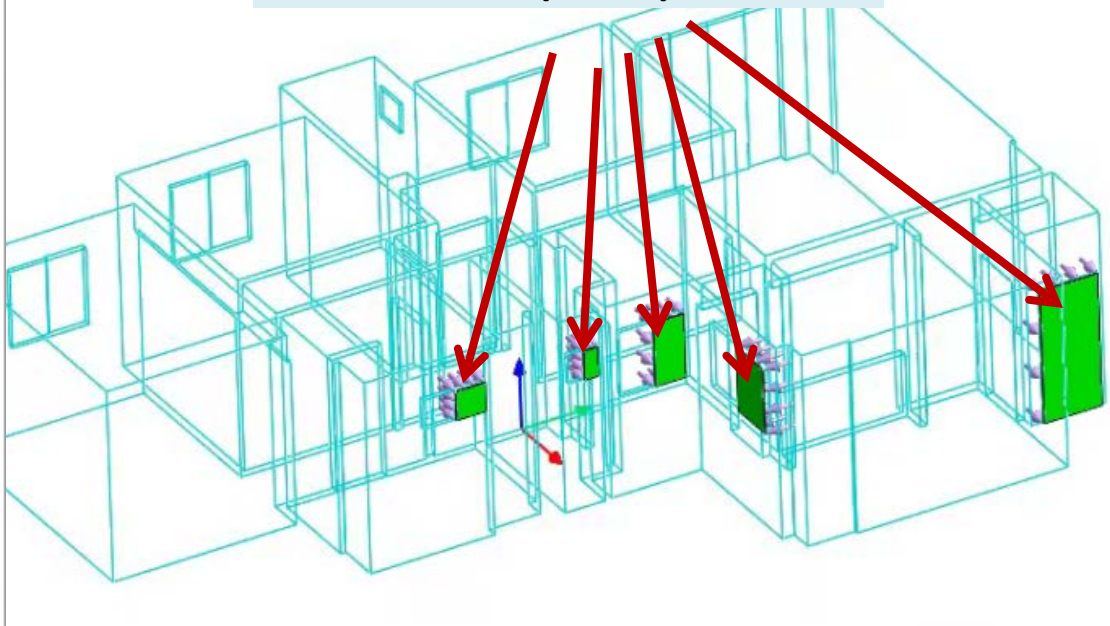
出流



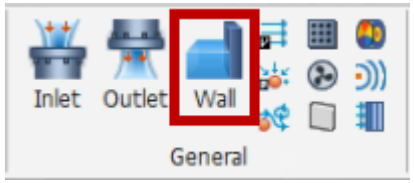


출流口特徵選取(自行判斷)

P=0(N/m²)

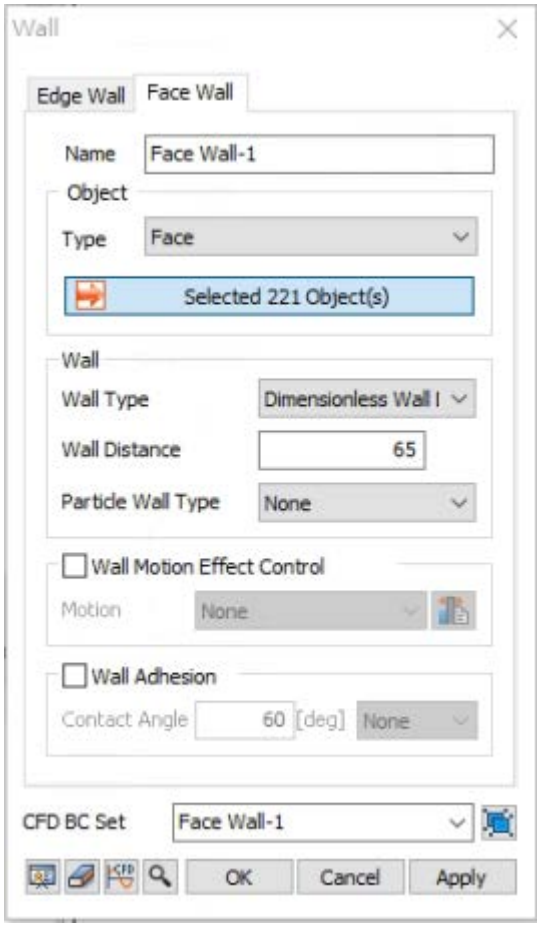


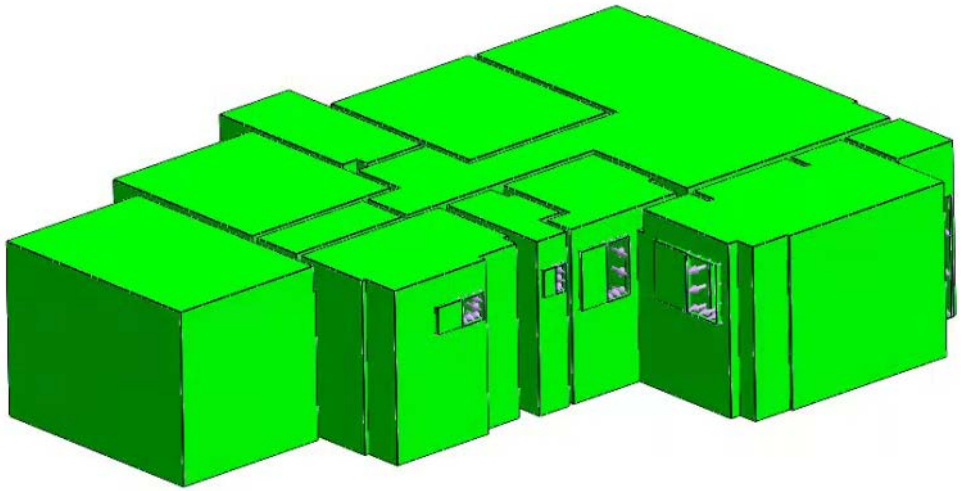
midas NFX-CFD



壁函數
(Wall)

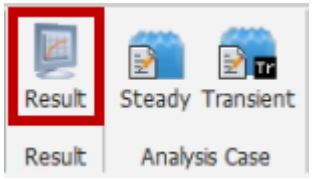
壁函數施加區域-流場和實體交界特徵
(Dimensionless Wall Distance y+=65)



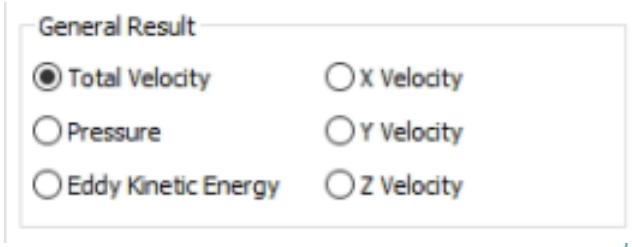


MIDAS

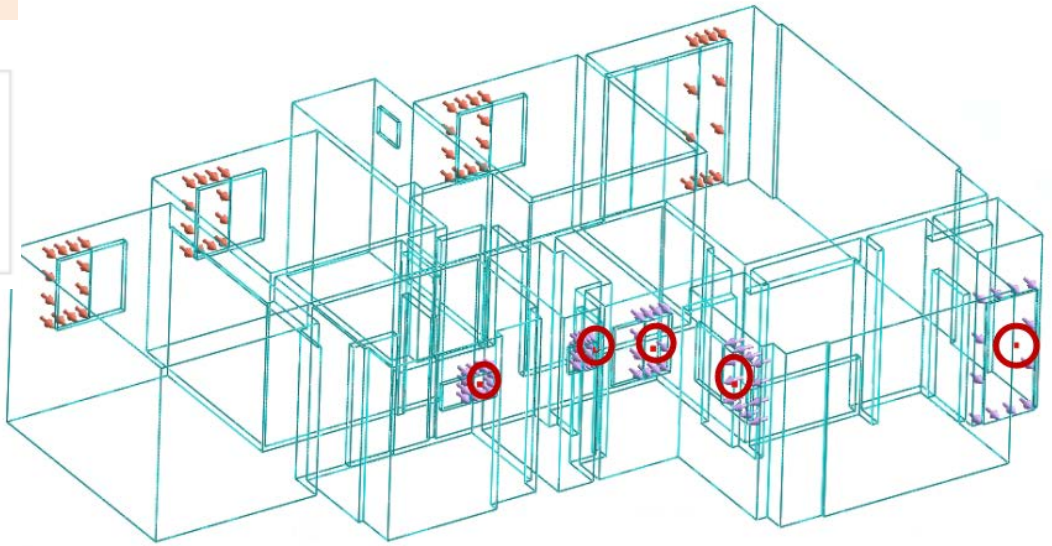
觀測點



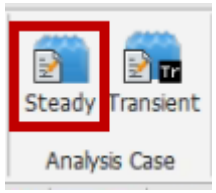
輸出特定節點(速度)數據



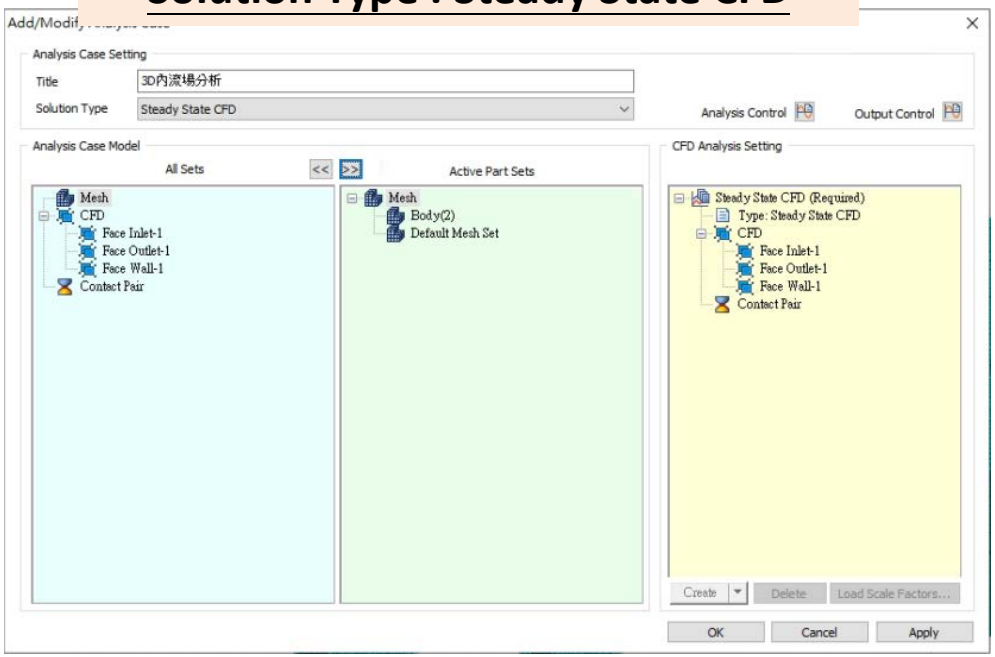
選取出流口節點作為觀測節點

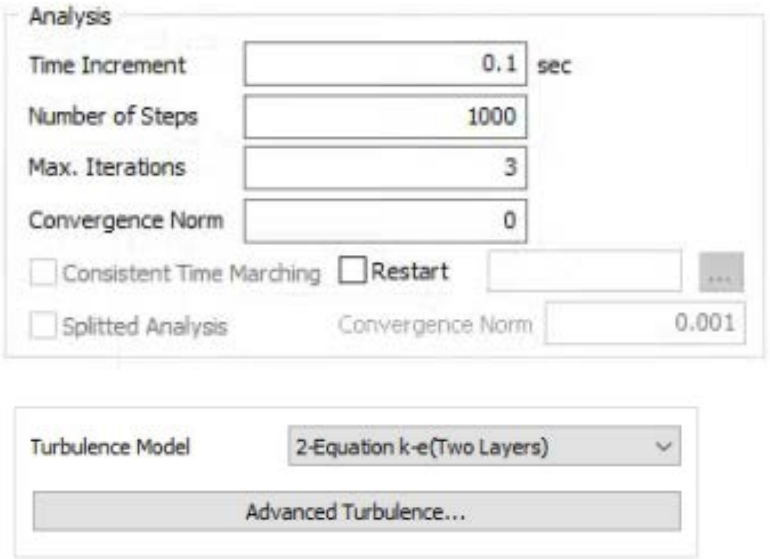


CFD穩態分析

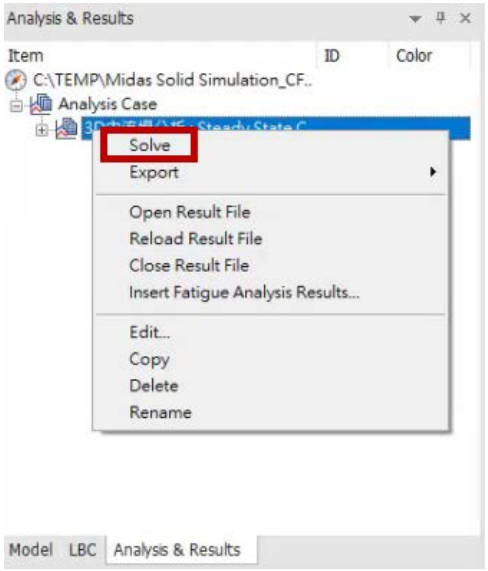


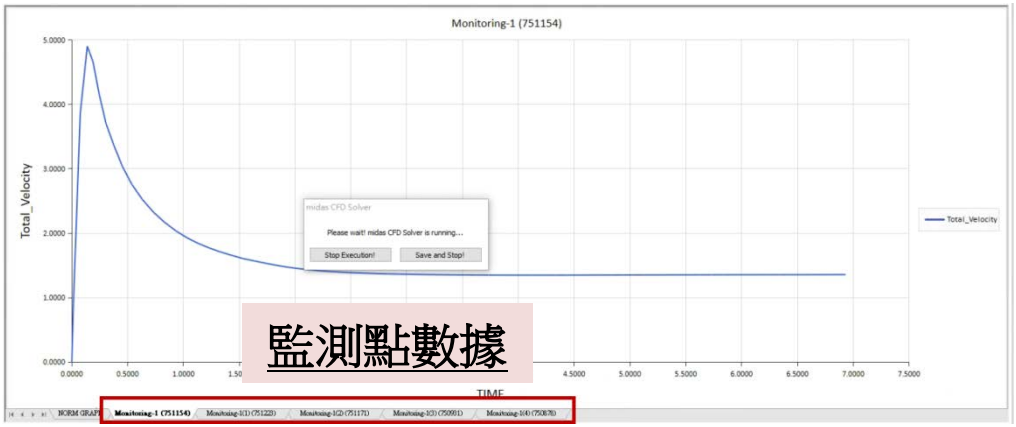
Solution Type : Steady State CFD



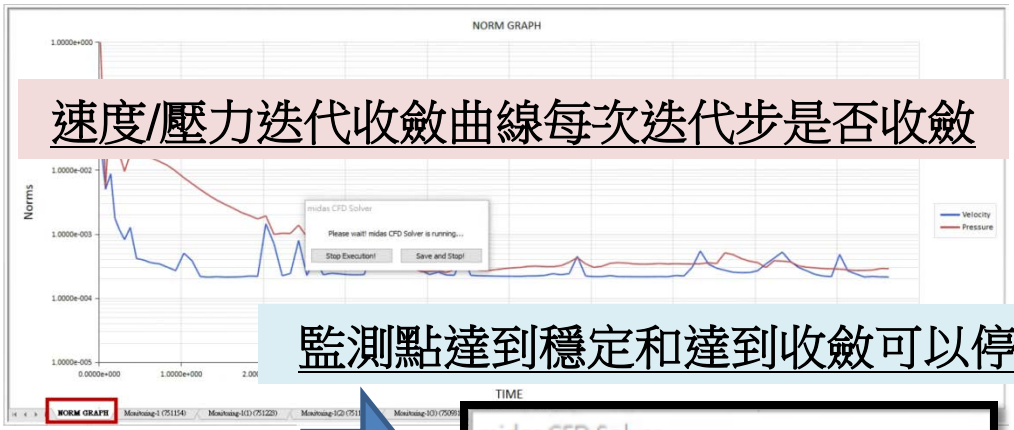


執行CFD求解





監測點數據



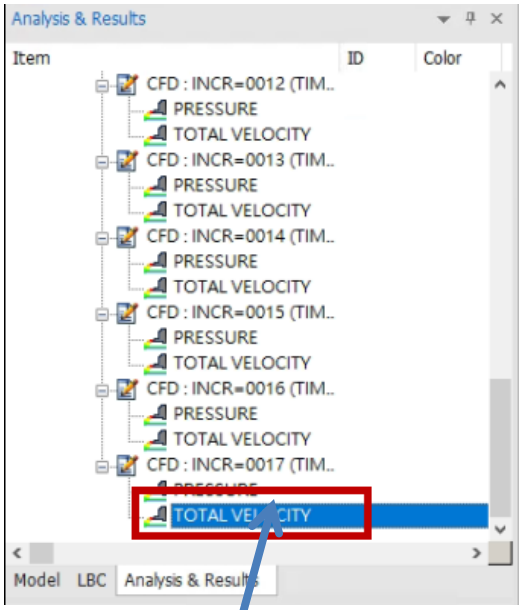
速度/壓力迭代收斂曲線每次迭代步是否收斂

監測點達到穩定和達到收斂可以停止計算

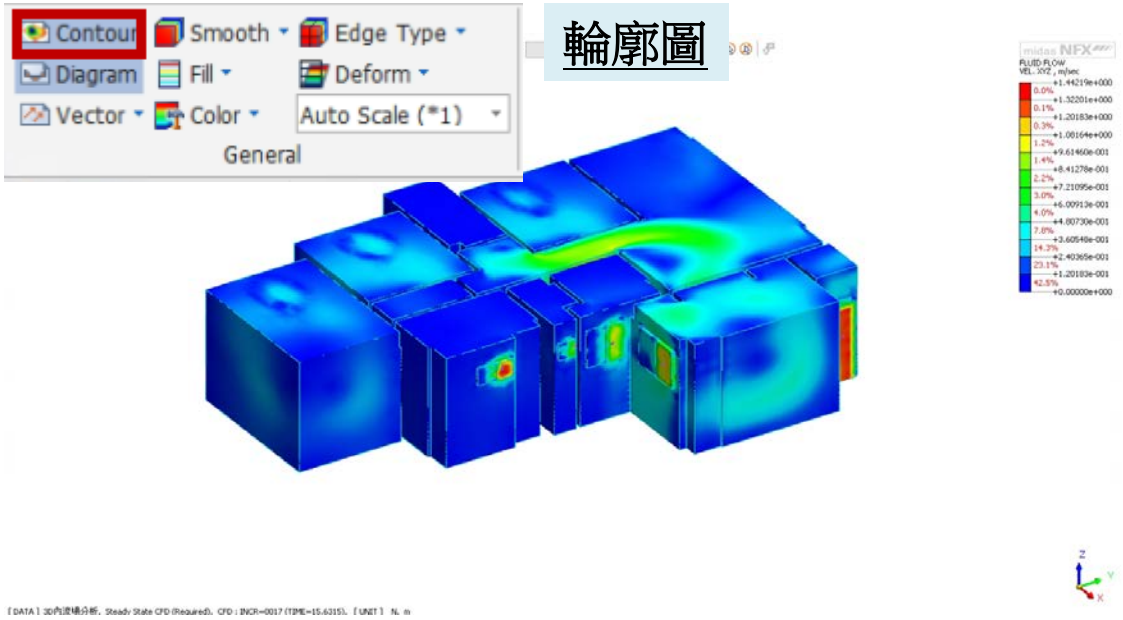
註1:透過收斂曲線和觀測點數據數據判斷是否斂

註2:如果無法收斂和達到穩定,判斷邊界是否合理和加密網格

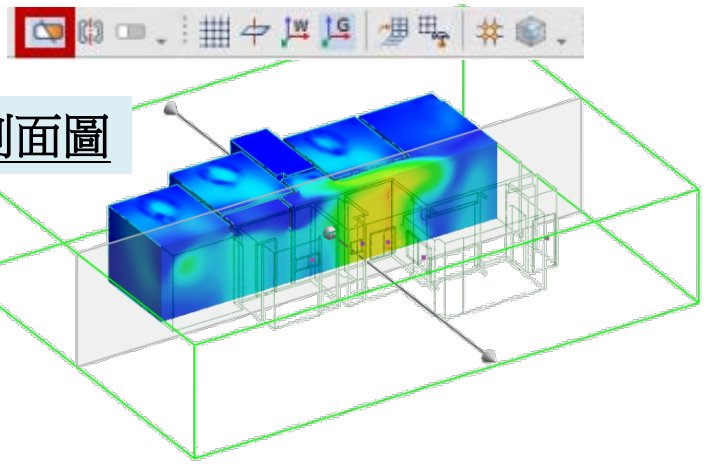
分析結果-1



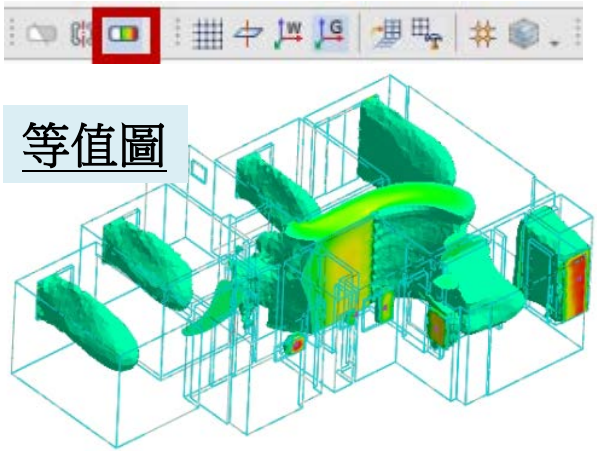
選擇最後一增量流速結果



輪廓圖

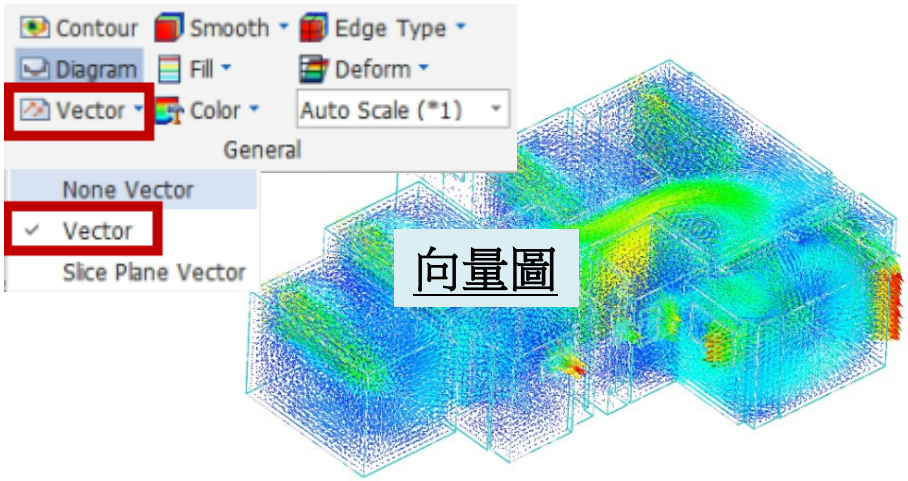


剖面圖

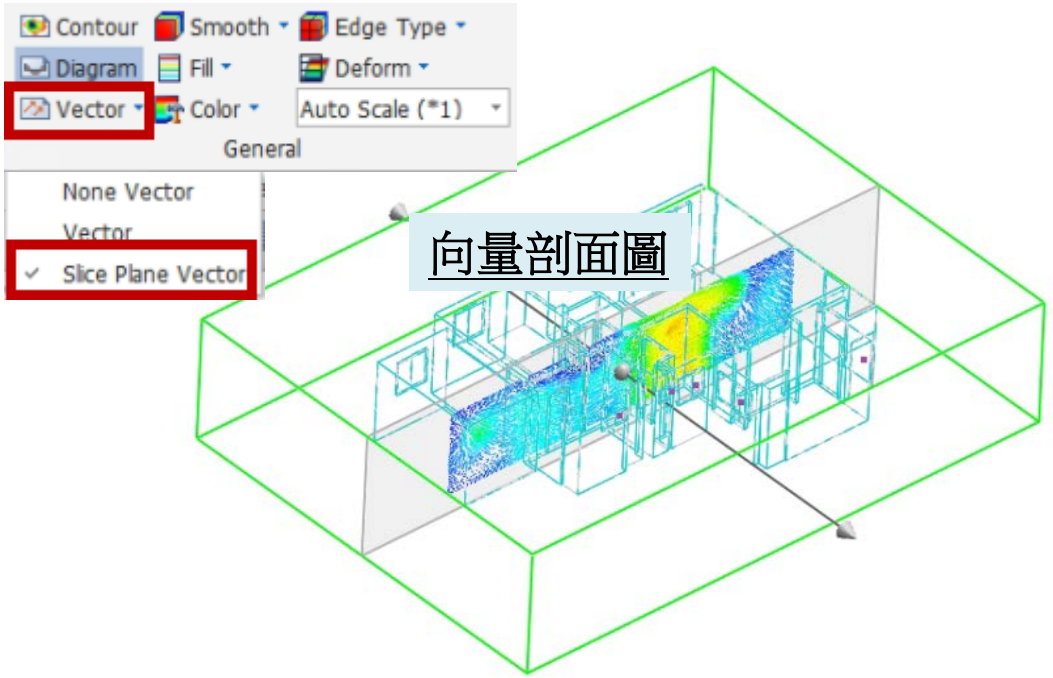


等值圖

分析結果-2

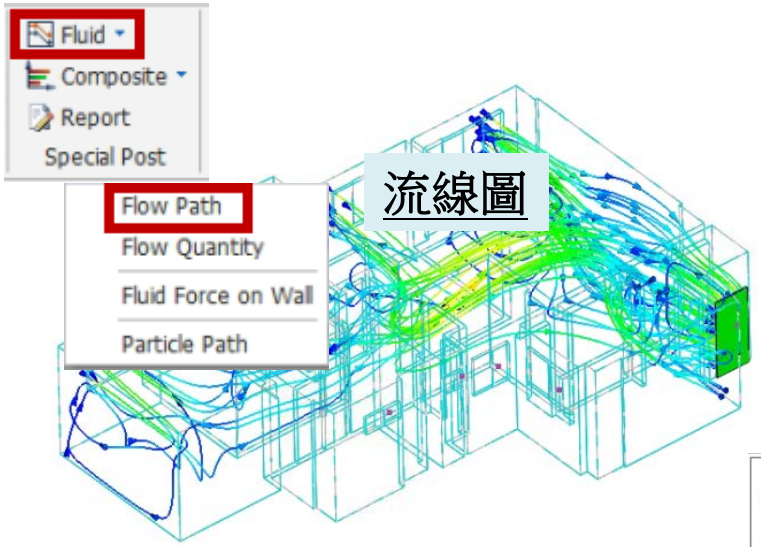


向量圖

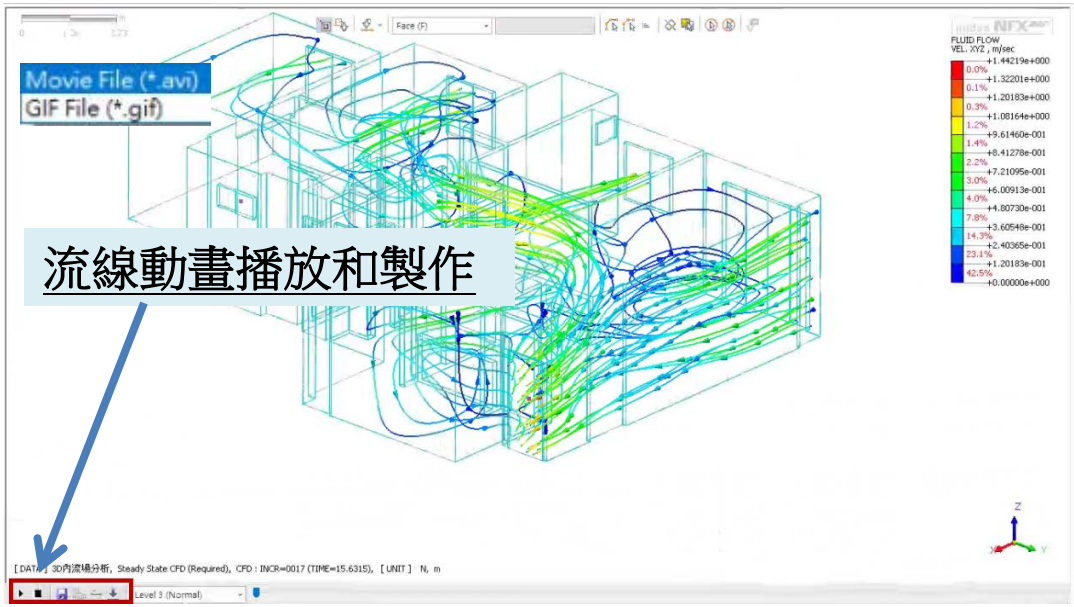


向量剖面圖

分析結果-3



流線圖



流線動畫播放和製作

CFD_標準教學系列

強制對流分析102-3D外流場分析

台灣邁達斯

風廓線
(Wind Speed Profile)

風廓線即風速隨高度的變化曲線，以研究大氣邊界層內的風速規律。常用邊界層風速剖面的有兩定律，一為指數律(power law)，另一則為對數律(logarithmic law)。

指數律風廓線
(Wind profile power law)

$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

u:is the wind speed(m/sec) at height z (m)
u_γ:is the known wind speed(m/sec) at a reference height z_γ (m)
α : is an empirically derived coefficient that varies dependent upon the stability of the atmosphere.For neutral stability conditions , α is approximately 1/7 。

對數風廓線
(Log wind profile)

$$u_z = \frac{u_*}{\kappa} \left[\ln \left(\frac{z-d}{z_0} \right) \right]$$

u_z:is the wind speed(m/sec) at height z (m)
u_* : is the friction velocity (m/s)
k: is the Von Kármán constant (~0.41)
d : is the zero plane displacement (m)
Z_0:is the surface roughness (m)

Reference
https://en.wikipedia.org/wiki/Wind_profile_power_law
https://en.wikipedia.org/wiki/Log_wind_profile

風廓線CFD函數定義
(指數律風廓線)

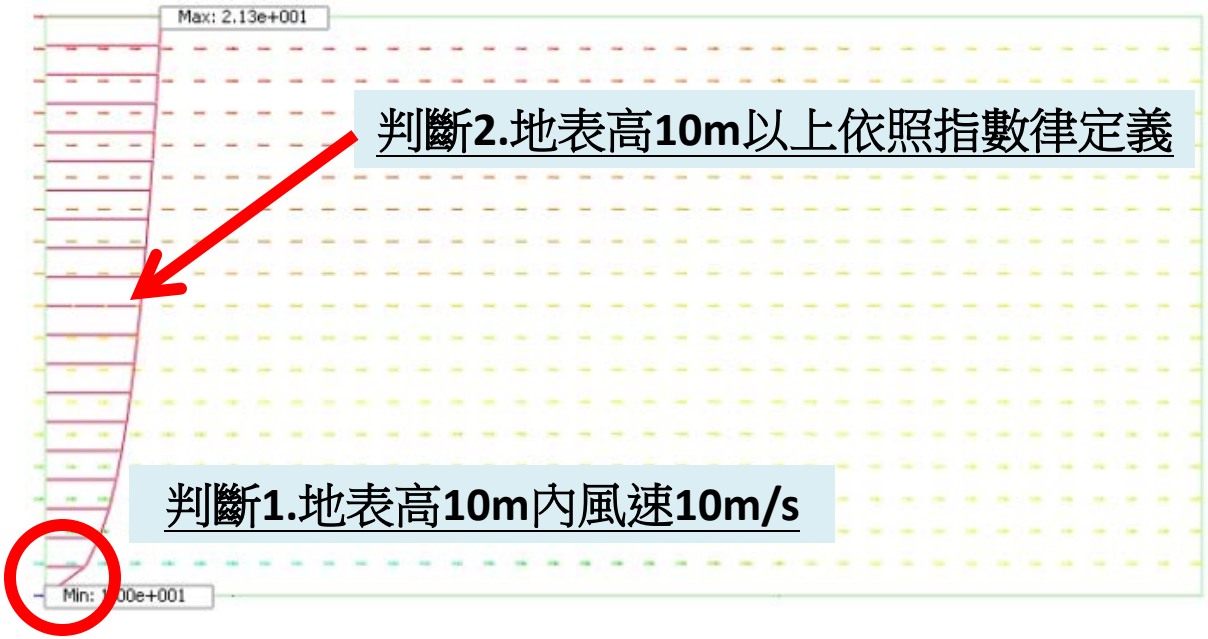
$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

u:相對高度風速(m/s)
Z_γ:地表高度(10m)
u_γ:地表高度Z_γ風速(假設10m/s)
α :0.143

CFD IF Function格式

if(expression)then(elif_expression)else(next_expression)endif

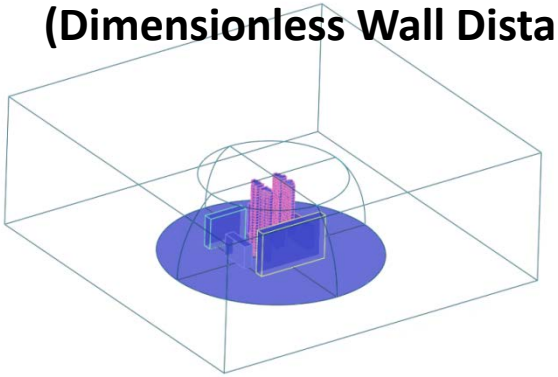
=>if(z<10)then(10)else(10*(z/10)^(0.143))endif



註:3D空間高度座標為z軸。

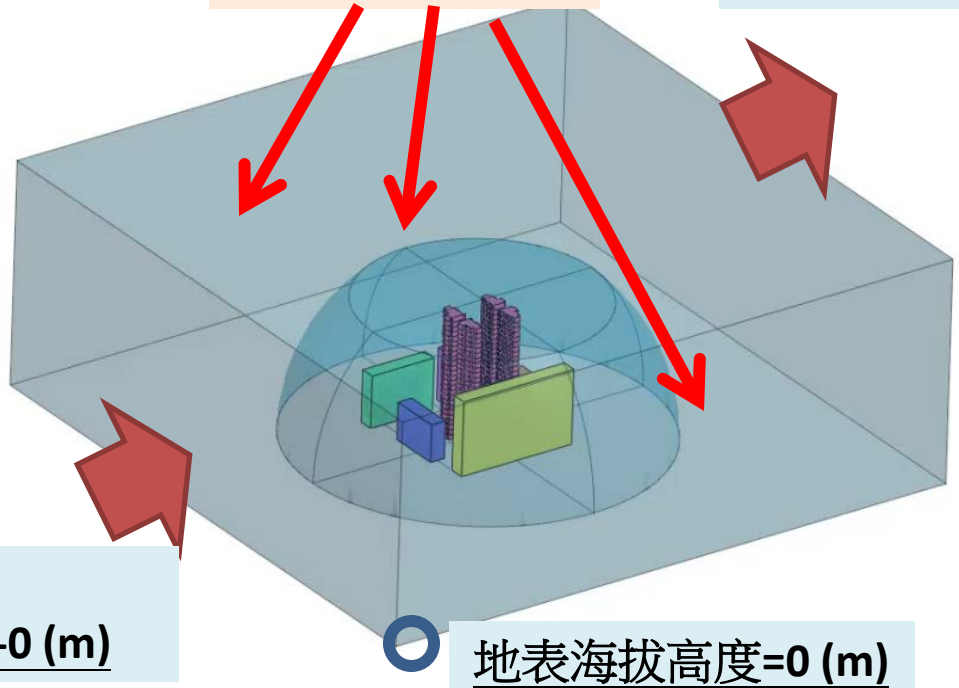
分析說明

壁函數施加區域-流場和實體交界特徵
(Dimensionless Wall Distance y+=65)



空間特徵-對稱

壓力0(N/m²)

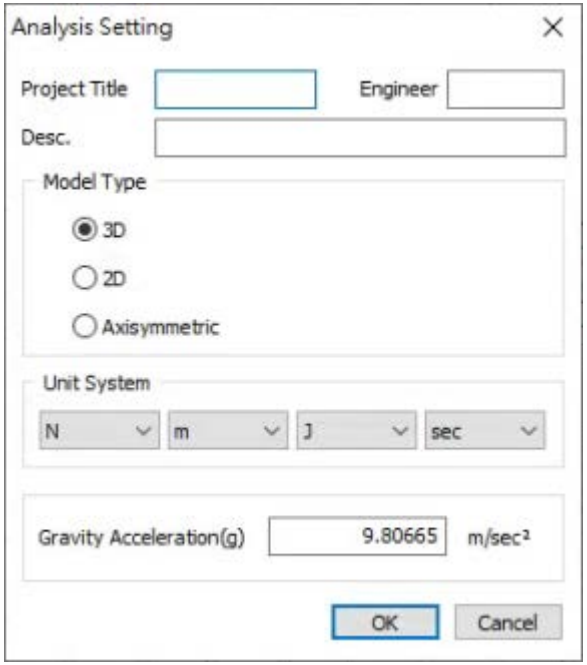


入流
海拔高度z-0 (m)

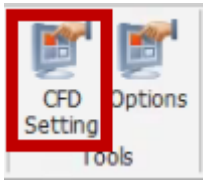
地表海拔高度=0 (m)

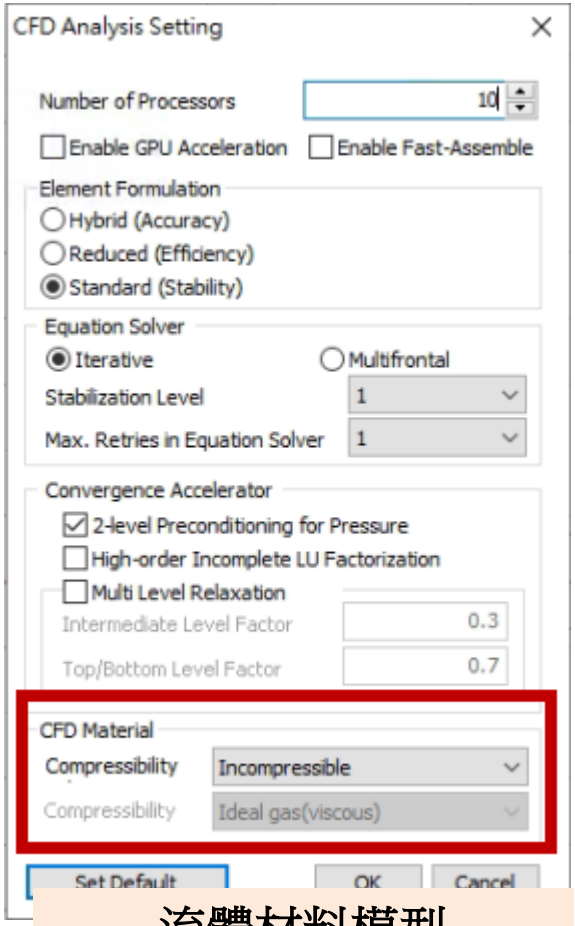
入流流速-函數定義
if(z<(10))then(10)else(10*((z)/10)^(0.143))endif

環境



單位使用N/m/J/sec

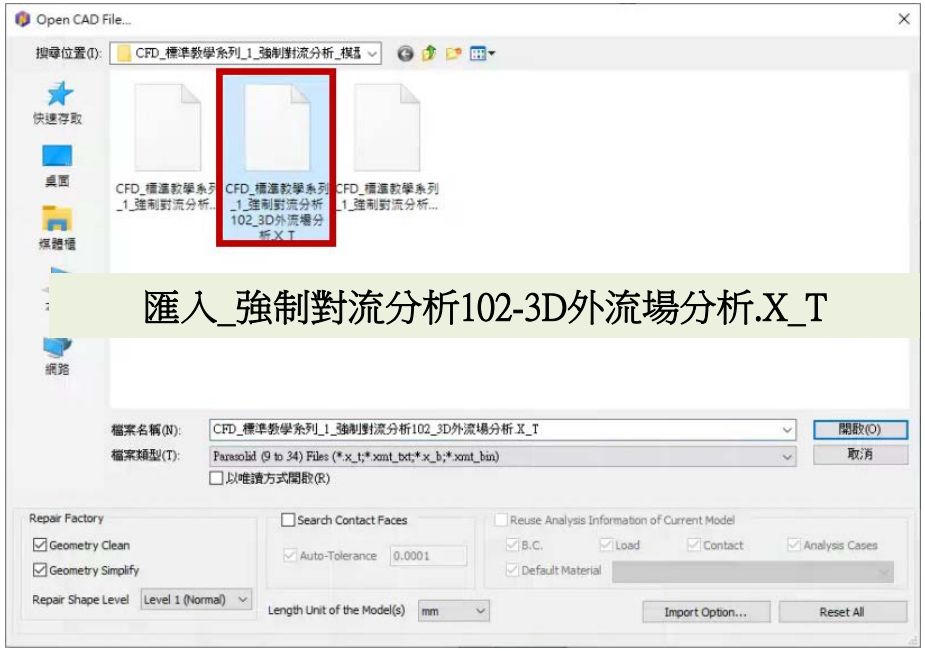


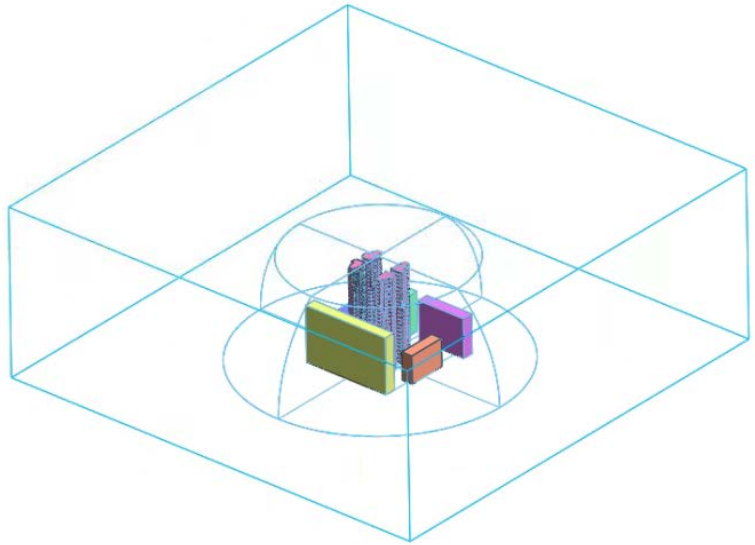


流體材料模型
不可壓縮流

匯入模型

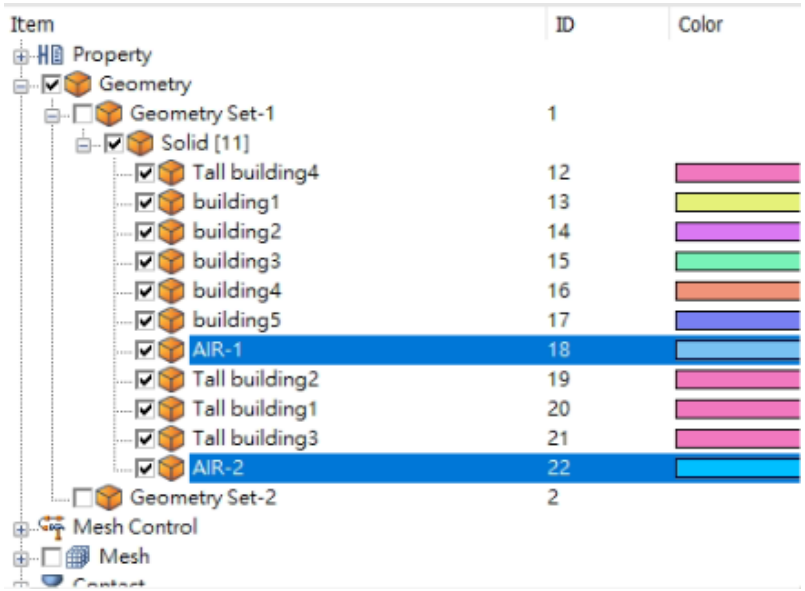

CAD
CAD
Import
Export
CAD File

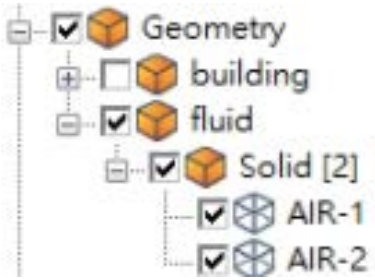

Open CAD File...
搜尋位置(I):
CFD_標準教學系列_1_強制對流分析_模型
快速存取
桌面
模體檔
網路
CFD_標準教學系列_1_強制對流分析102_3D外流場分析.X_T
匯入_強制對流分析102-3D外流場分析.X_T
檔案名稱(N):
CFD_標準教學系列_1_強制對流分析102_3D外流場分析.X_T
檔案類型(T):
Parasolid (9 to 34) Files (*.x_t;*.xmt_txt;*.x_b;*.xmt_bin)
開啟(O)
取消
以唯讀方式開啟(R)
Repair Factory
Geometry Clean
Geometry Simplify
Repair Shape Level
Level 1 (Normal)
Search Contact Faces
Auto-Tolerance
0.0001
Reuse Analysis Information of Current Model
B.C.
Load
Contact
Analysis Cases
Default Material
Length Unit of the Model(s)
mm
Import Option...
Reset All

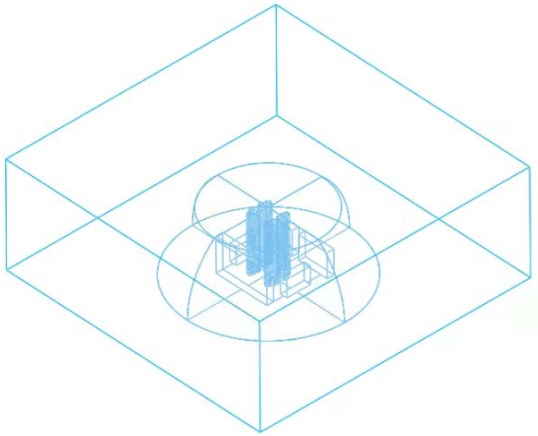


幾何集分類

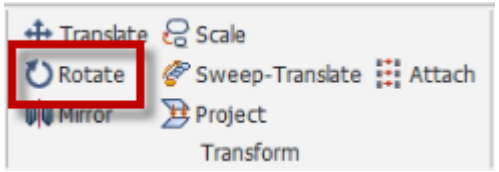
流場區域單獨定義成幾何集



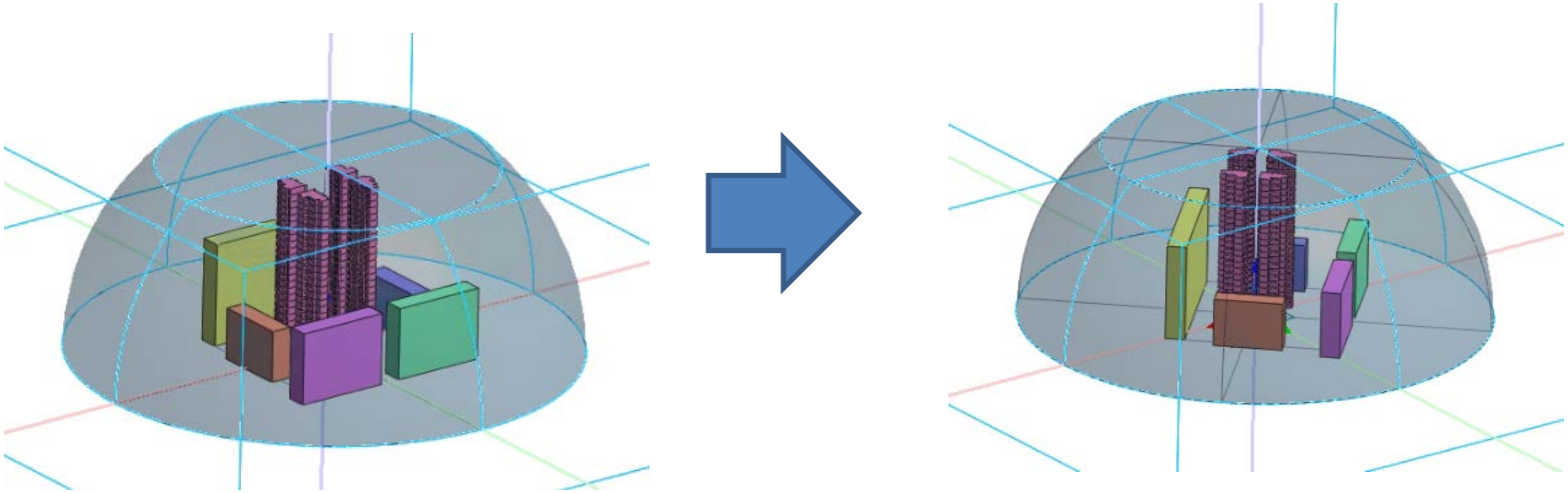




模型方位調整說明



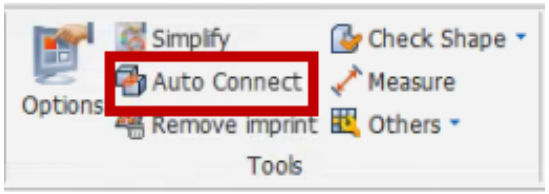
仿照風洞實驗,半圓模型建模,可以快速調整風位方向



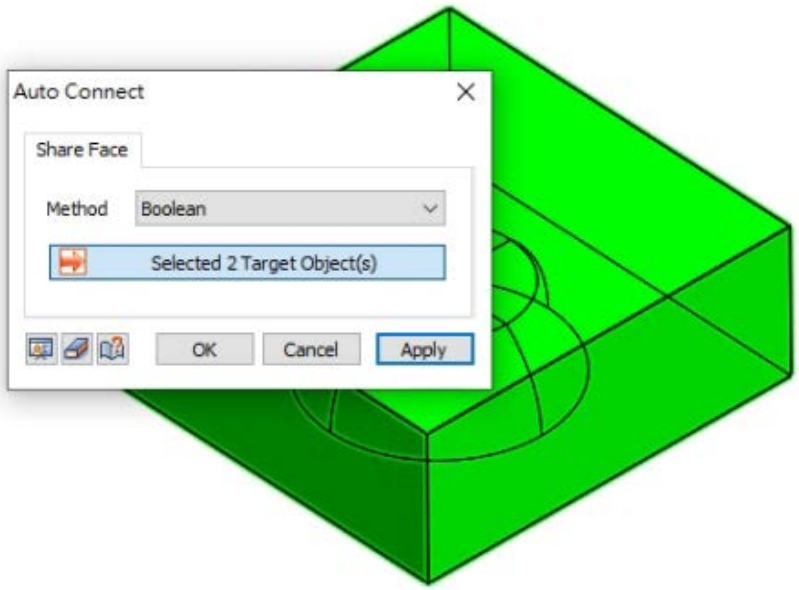
註1:範例不操作此分析步
註2:實際分析模型建議入流&出流方向10~15倍距離,高度和2側方向5~10倍距離。

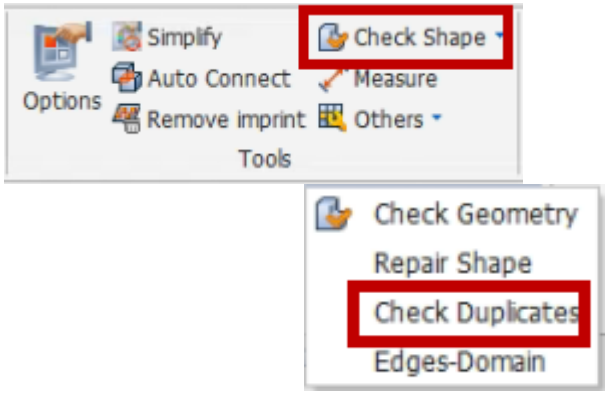
相鄰面特徵建立&確認

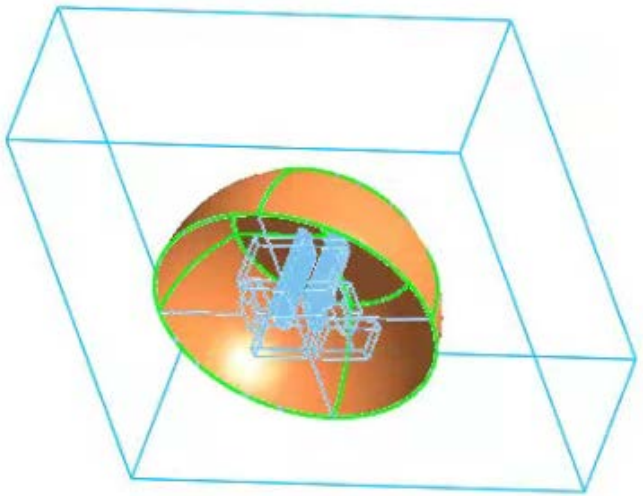
相鄰面特徵可產生共點位置網格



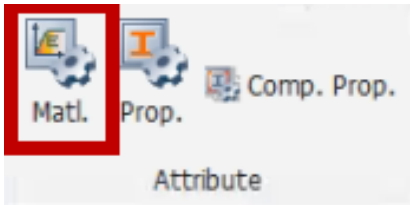
流場區域幾何特徵

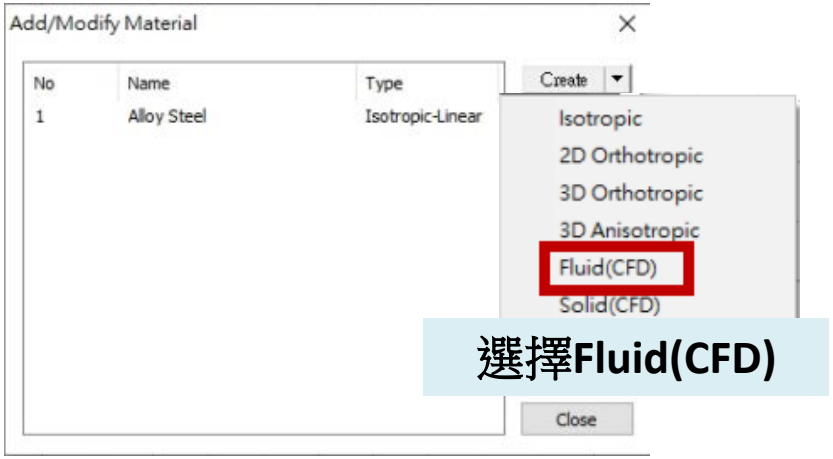






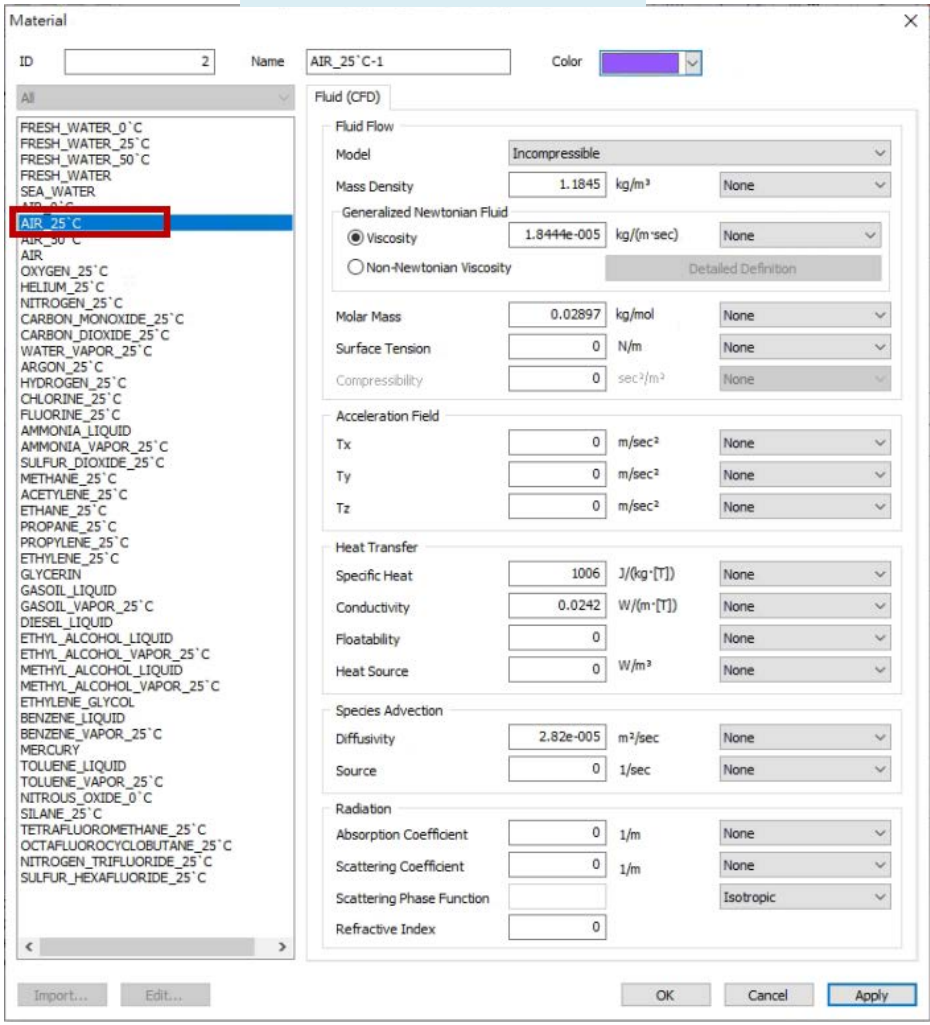
材料



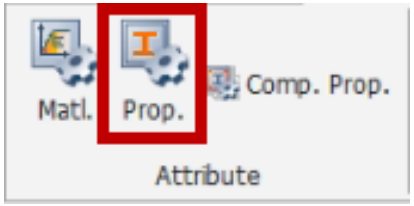


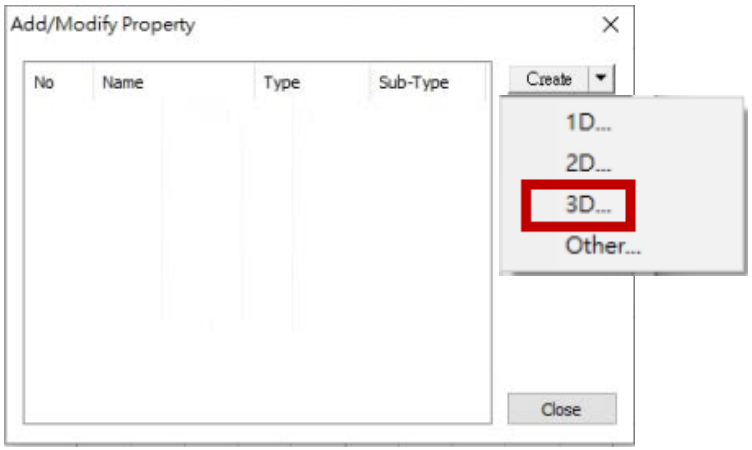
選擇Fluid(CFD)

新增AIR 25 ℃

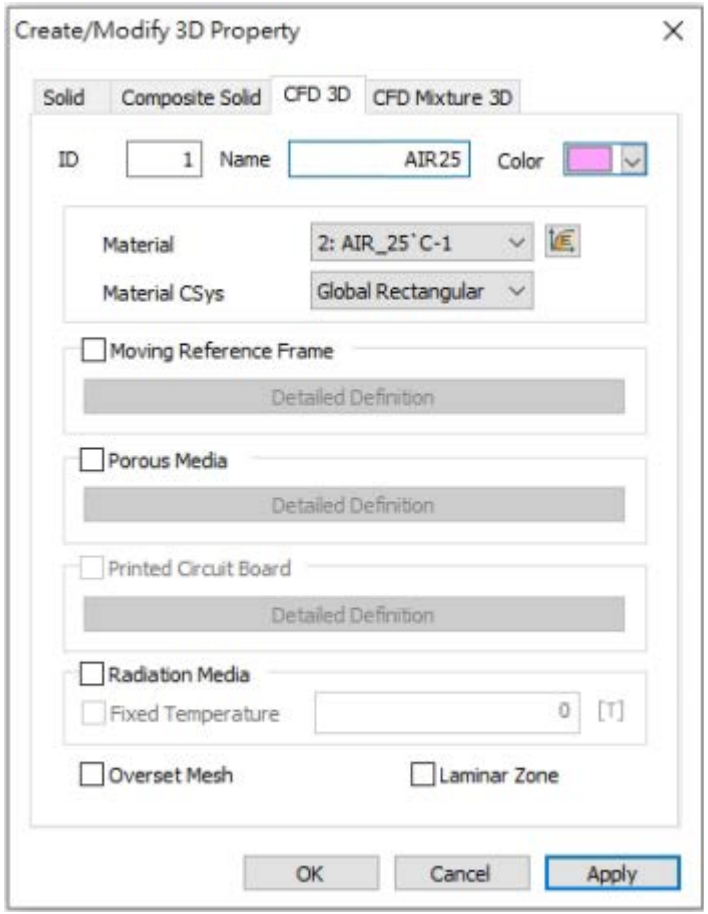


屬性

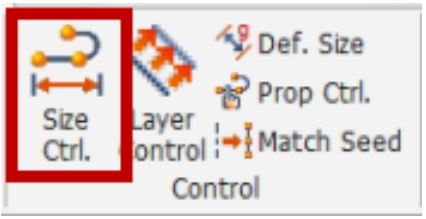




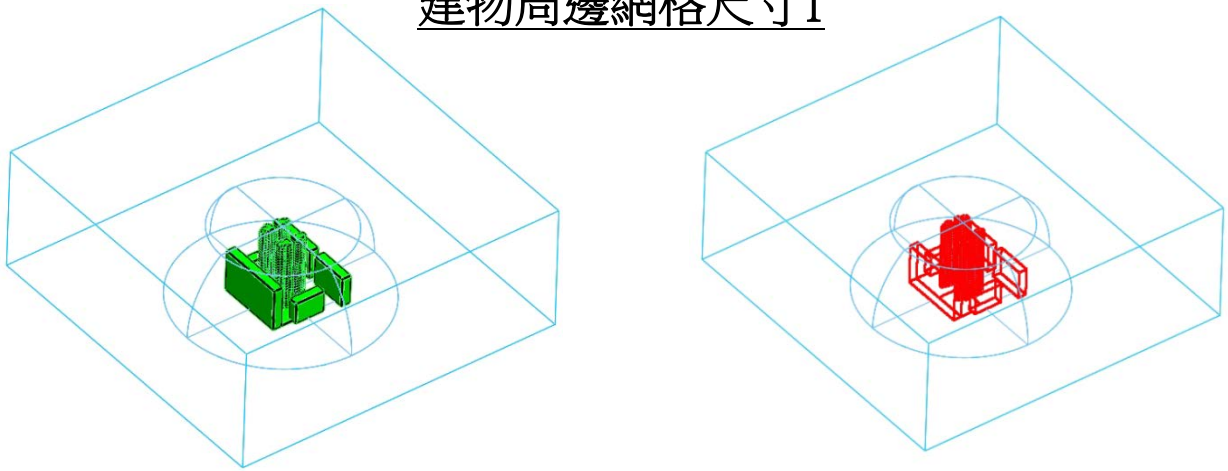
新增AIR 25 ℃ Property



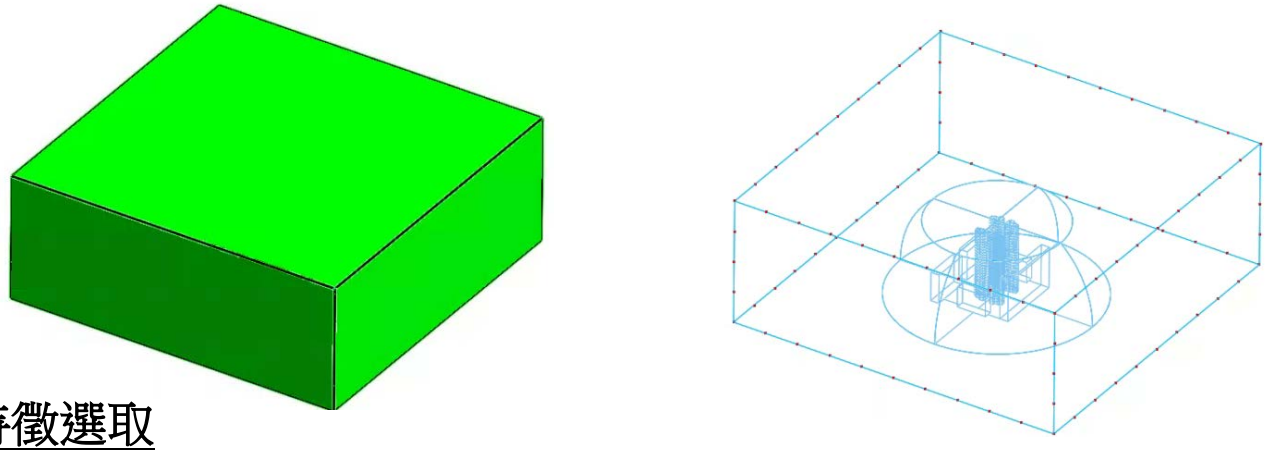
手動網格尺寸定義



建物周邊網格尺寸1



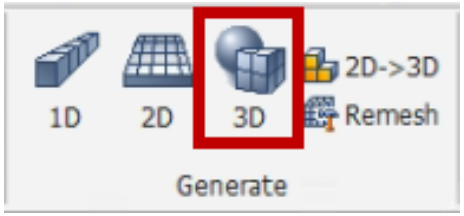
流場外圍網格尺寸40



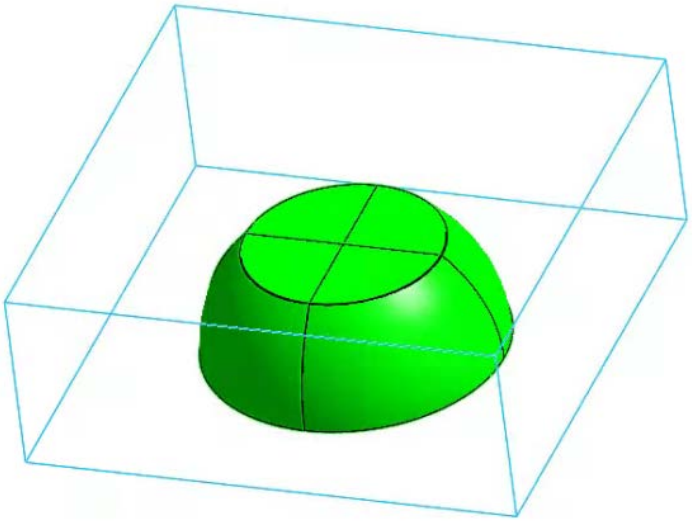
補充:建議使用面特徵選取

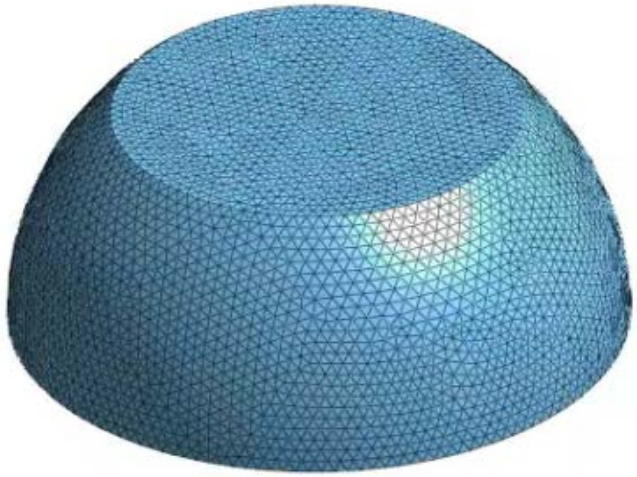
註:範例使用較粗糙網格尺寸，實際分析建議加密網格尺寸。

網格-建物周邊流場



Tetra Meshr(四面體網格)/網格尺寸5 (m)/1階元素

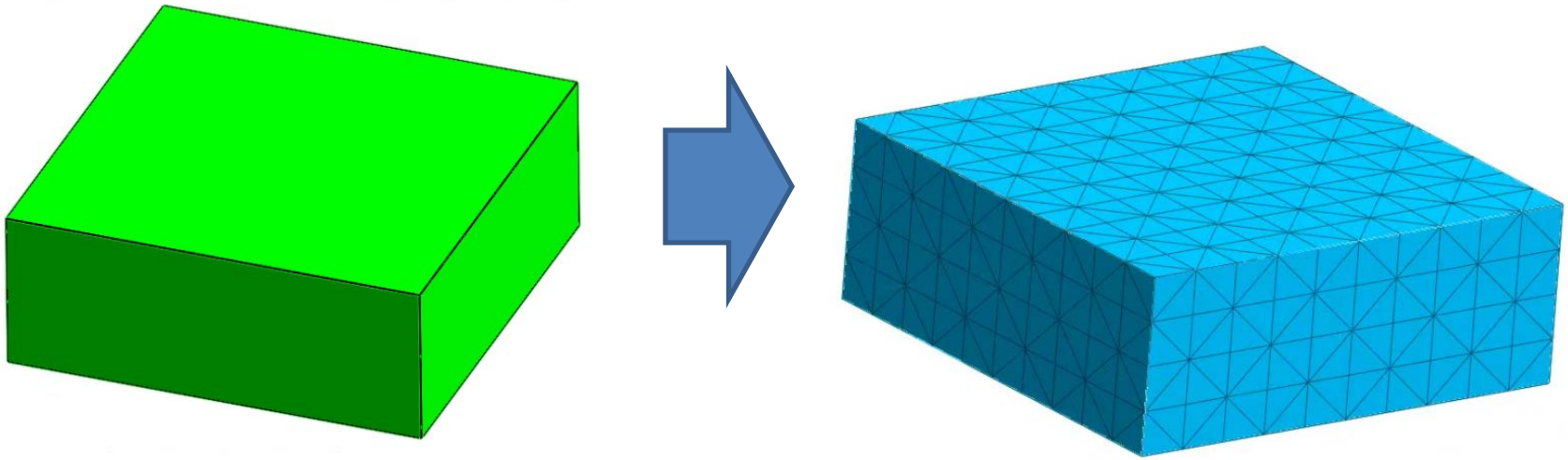




NFX CFD不支援二階(高階)元素

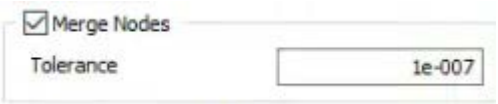
網格-外側流場

Tetra Meshr(四面體網格)/網格尺寸20 (m)/1階元素
相鄰特徵面/共節點

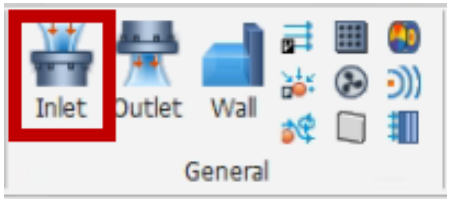


相鄰特徵面

合併節點

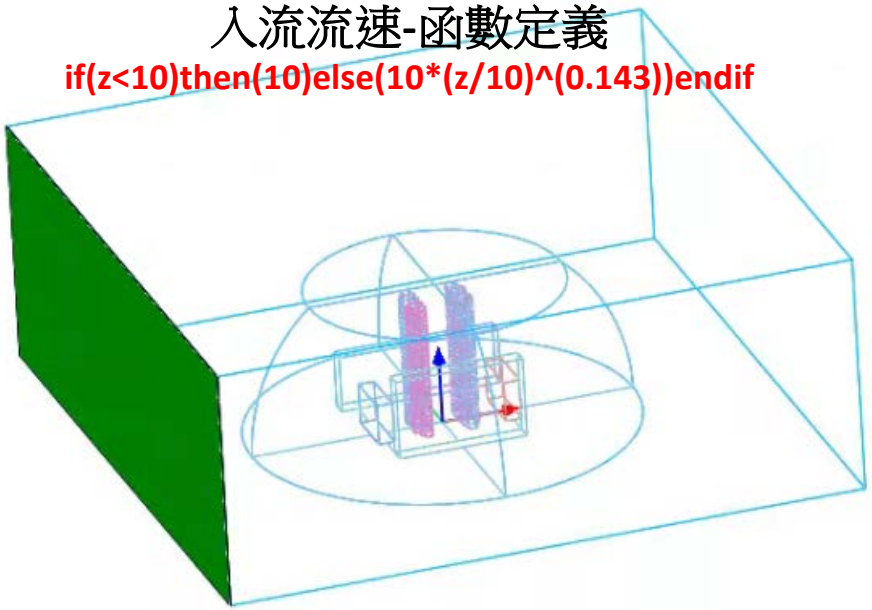


入流



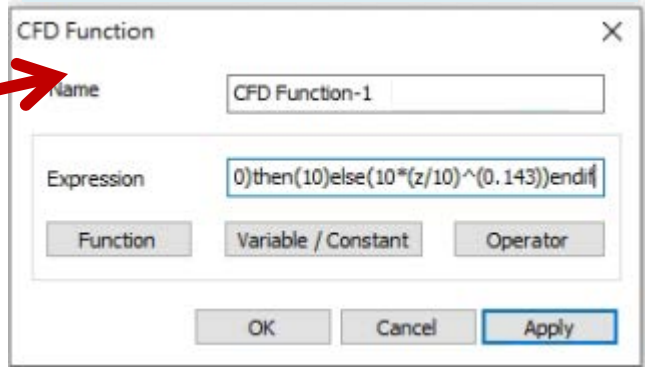
入流流速-函數定義

if(z<10)then(10)else(10*(z/10)^(0.143))endif



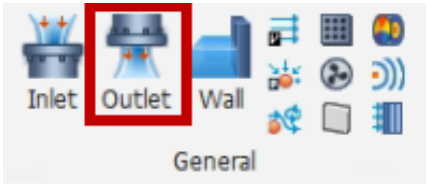
Step2.指定函數

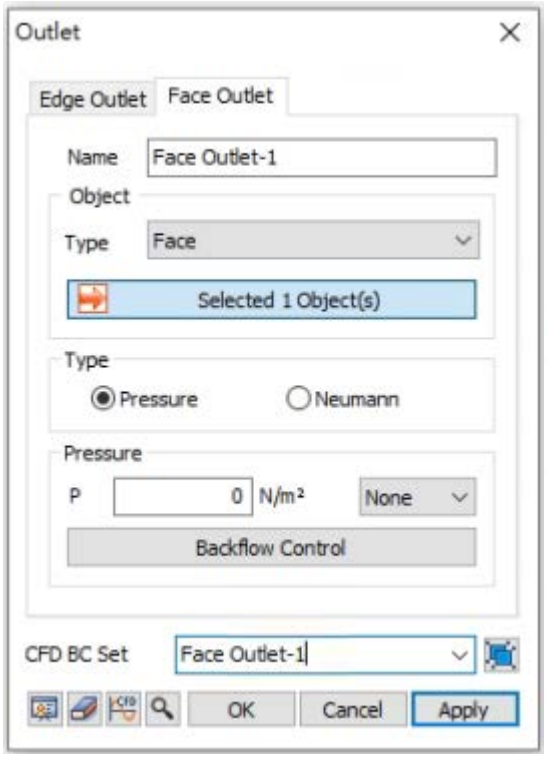
Step1.新增風廓線入流函數



註:不同邊界&載荷建議使用不同集合名稱。

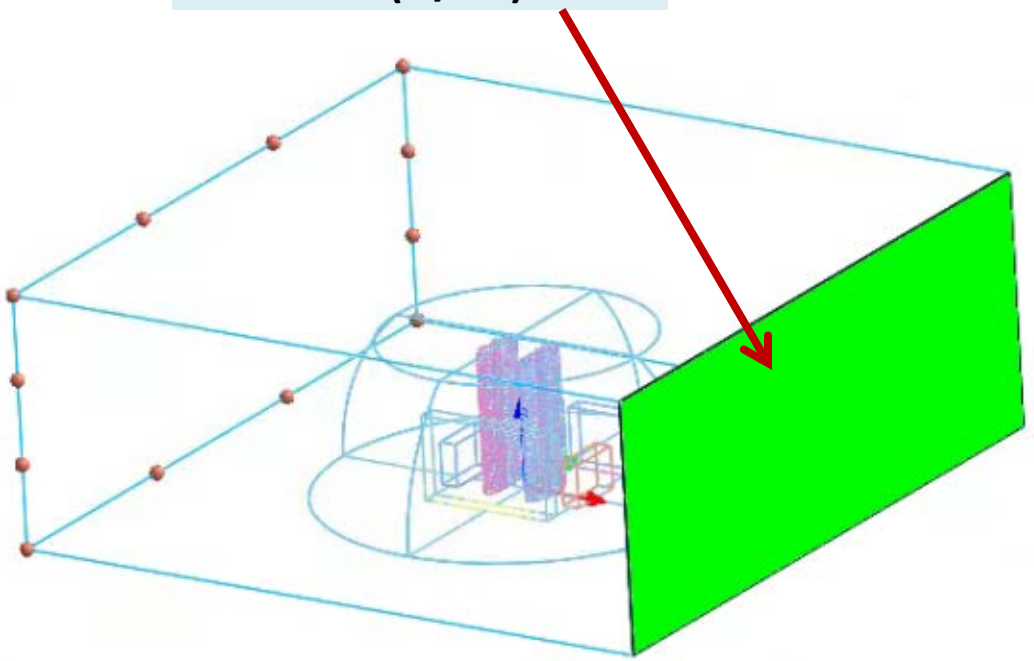
出流





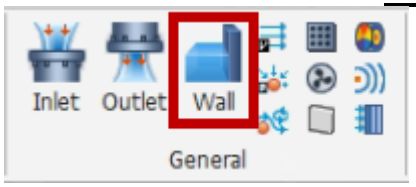
出流口特徵選取

P=0(N/m²)



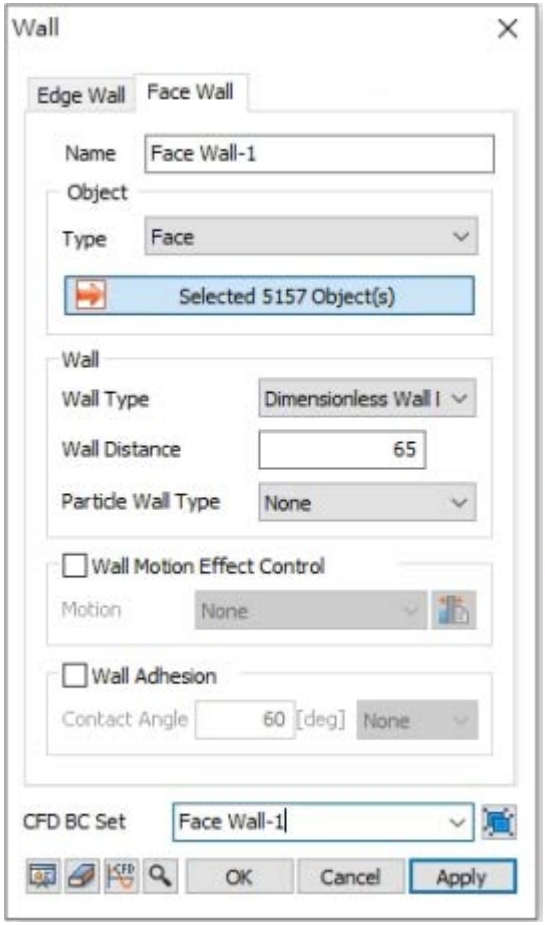
壁函數-建物和流場交界

(Wall)

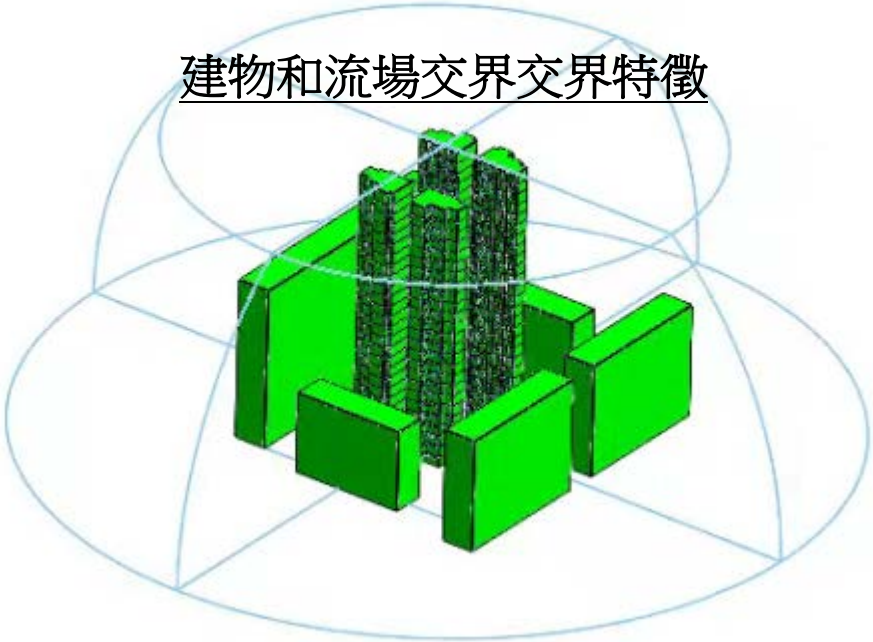


壁函數施加區域-流場和實體交界特徵

(Dimensionless Wall Distance y+=65)

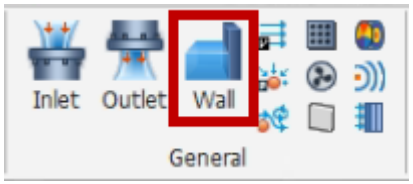


建物和流場交界交界特徵



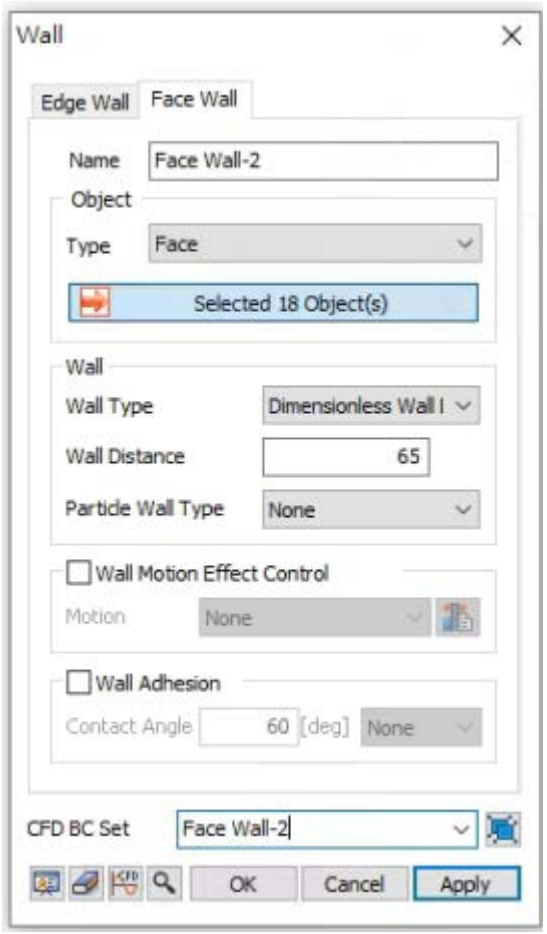
壁函數-地面和流場交界

(Wall)

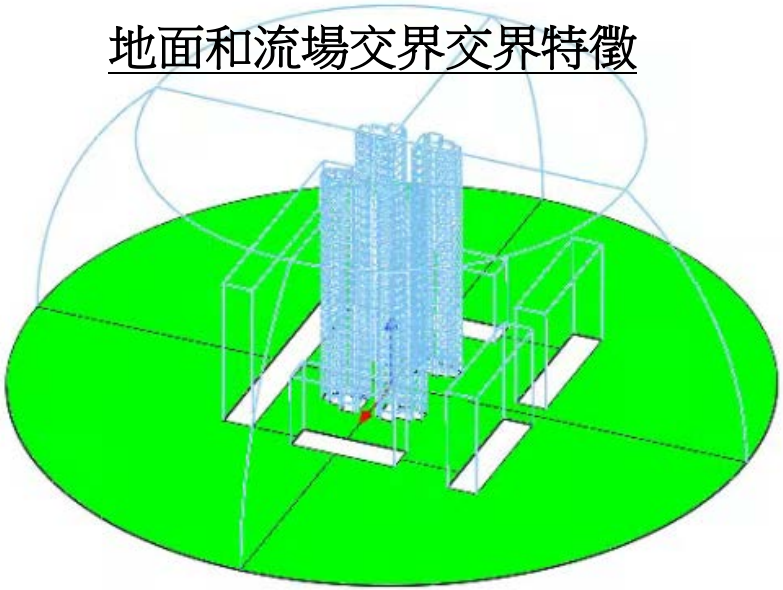


壁函數施加區域-流場和實體交界特徵

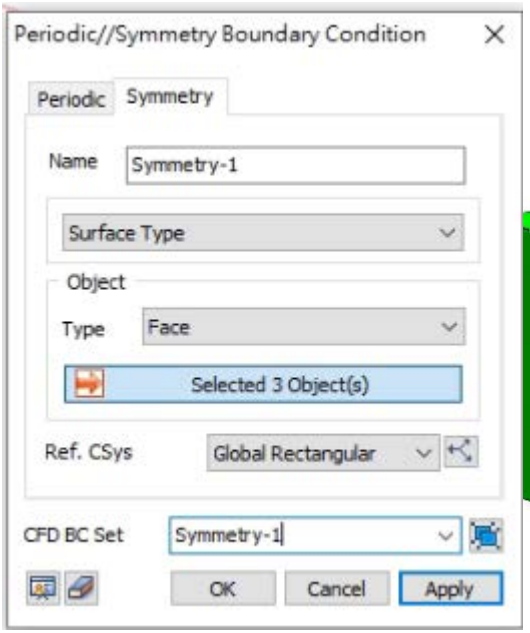
(Dimensionless Wall Distance y+=65)



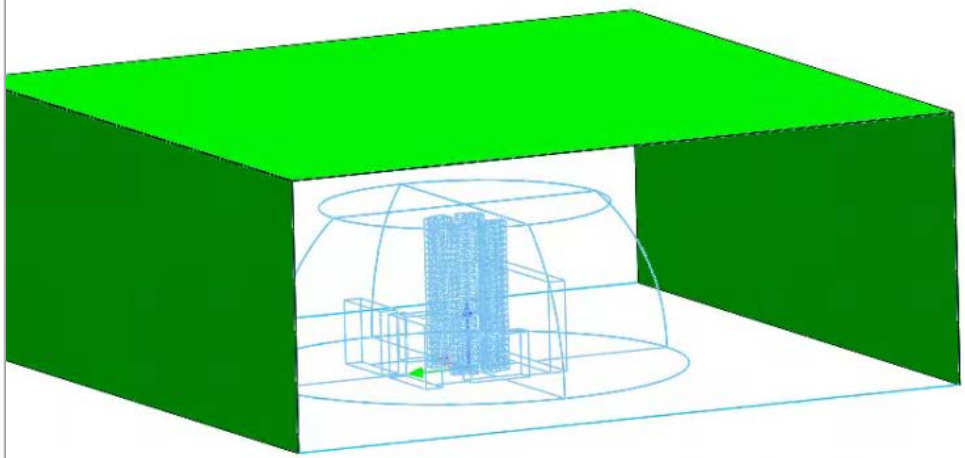
地面和流場交界交界特徵



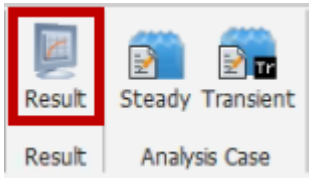
對稱
(Symmetry)



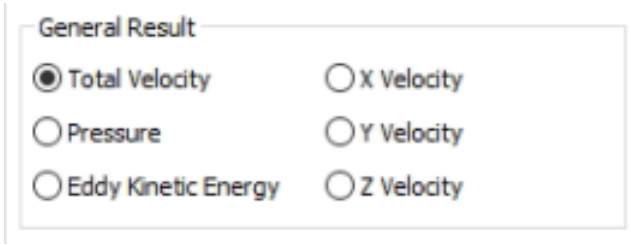
特徵選取
(非入流/出流/壁函數其它流場域特徵)



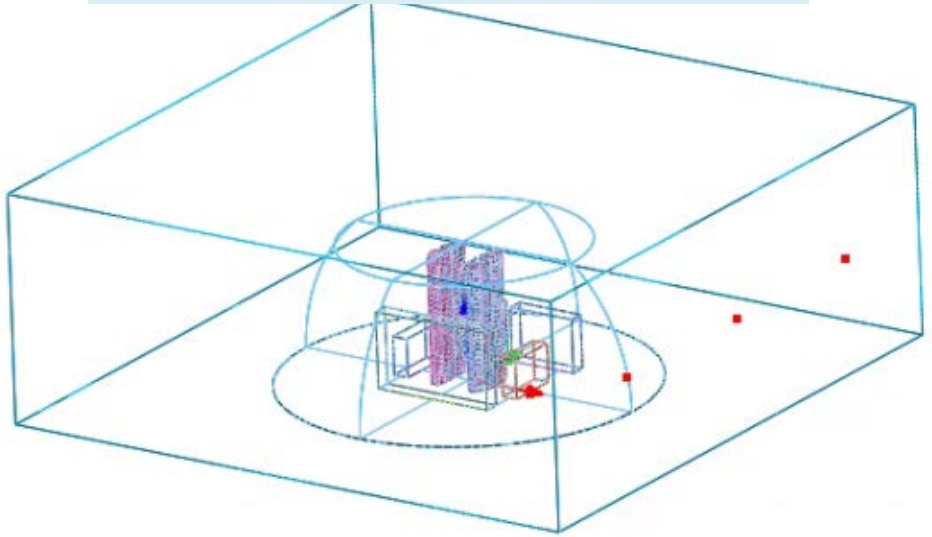
觀測點



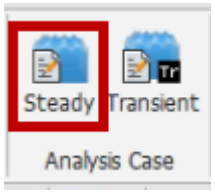
輸出特定節點(速度)數據



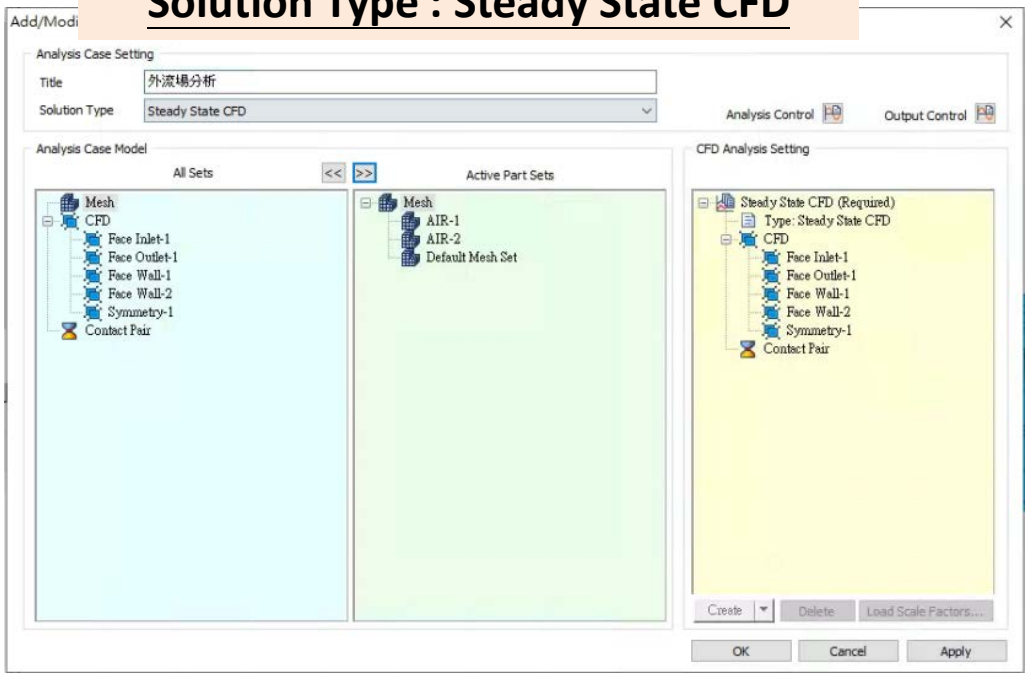
選取出流口節點作為觀測節點

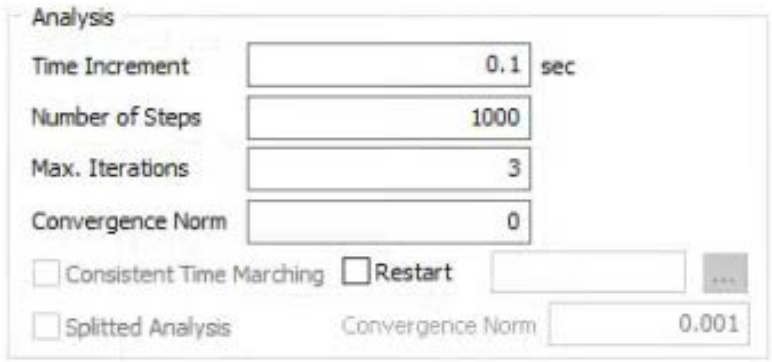


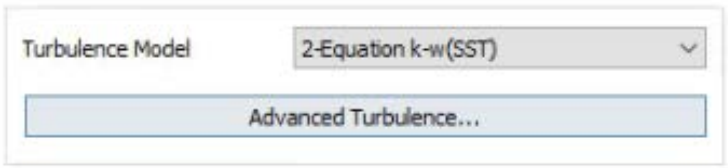
CFD穩態分析



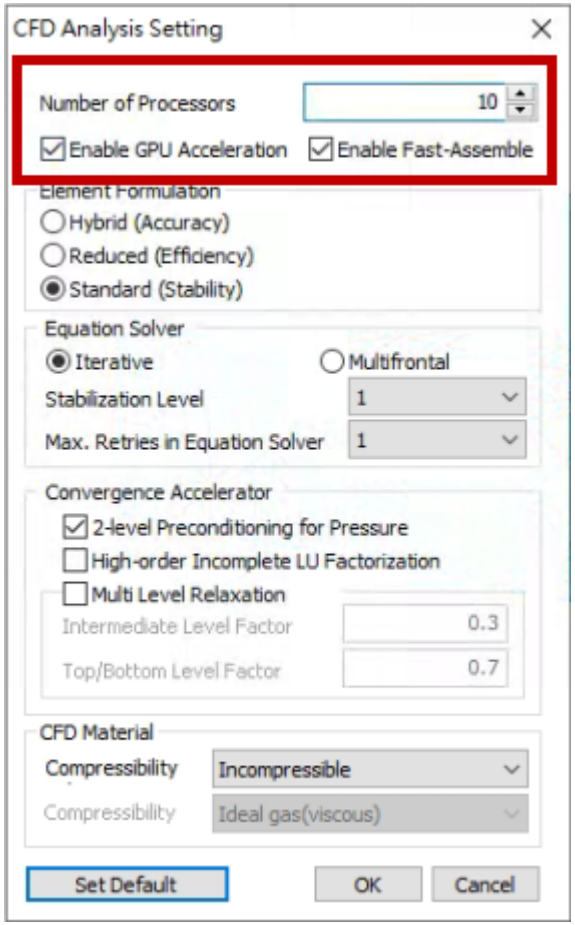
Solution Type : Steady State CFD





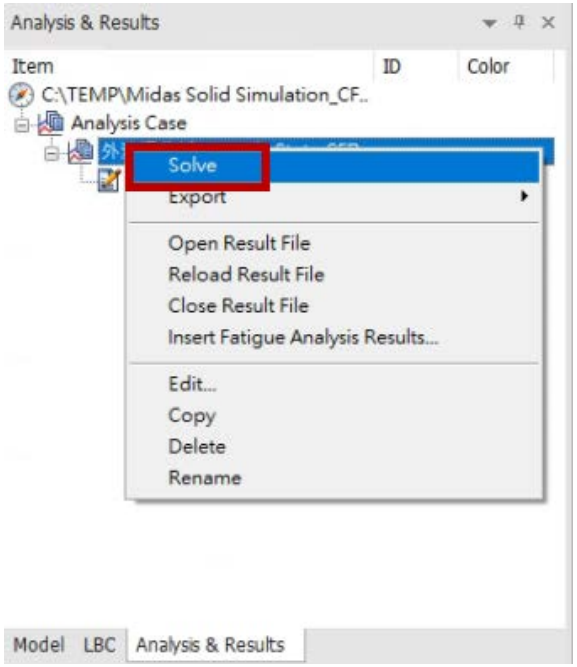


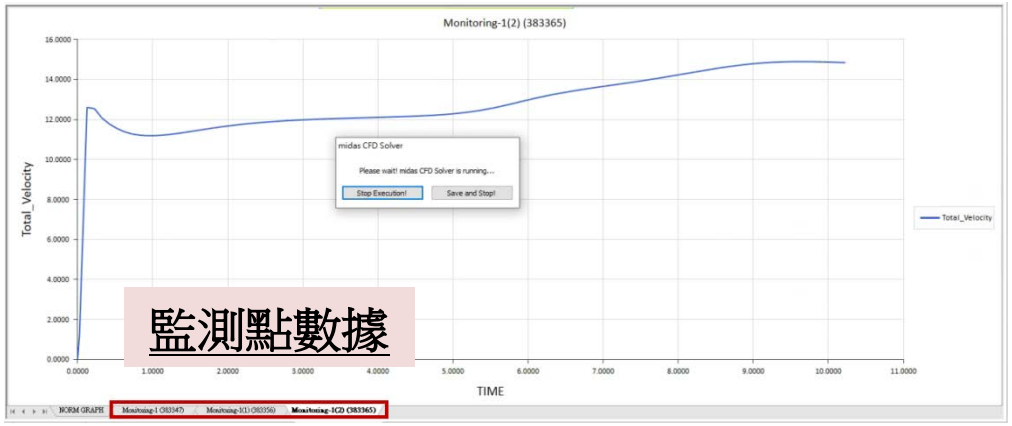
CPU&GPU定義


CFD Analysis Setting
Number of Processors
10
Enable GPU Acceleration
Enable Fast-Assemble
Element Formulation
Hybrid (Accuracy)
Reduced (Efficiency)
Standard (Stability)
Equation Solver
Iterative
Multifrontal
Stabilization Level
1
Max. Retries in Equation Solver
1
Convergence Accelerator
2-level Preconditioning for Pressure
High-order Incomplete LU Factorization
Multi Level Relaxation
Intermediate Level Factor
0.3
Top/Bottom Level Factor
0.7
CFD Material
Compressibility
Incompressible
Compressibility
Ideal gas(viscous)
Set Default
OK
Cancel

執行CFD求解

(自行終止計算)



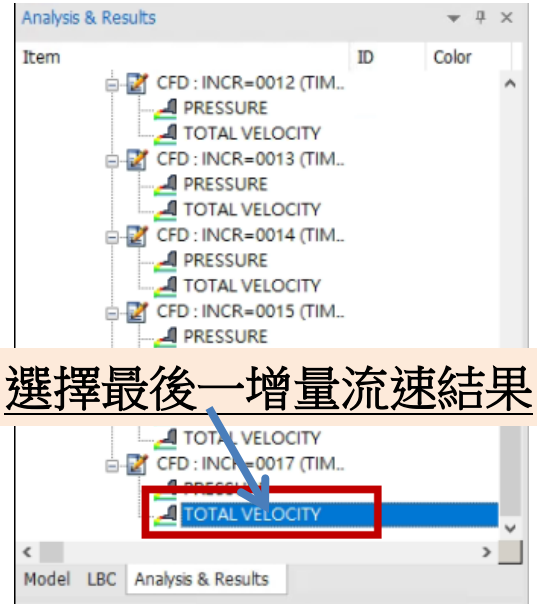


監測點達到穩定和達到收斂可以停止計算

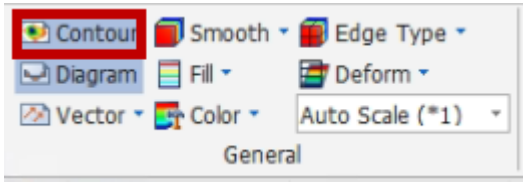
註1:透過收斂曲線和觀測點數據數據判斷是否斂

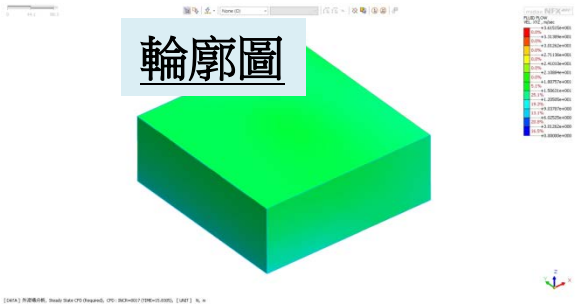
註2:如果無法收斂和達到穩定,判斷邊界是否合理和加密網格

分析結果-1

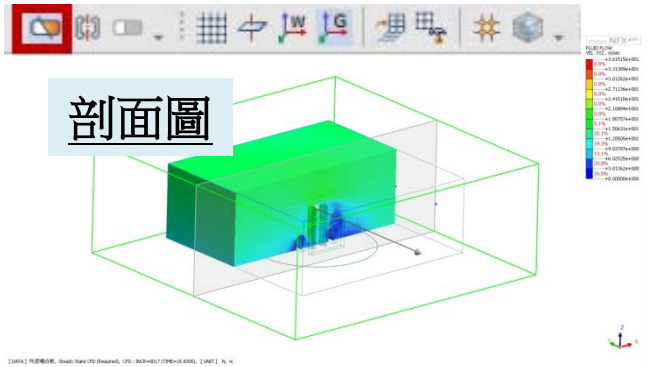


選擇最後一增量流速結果



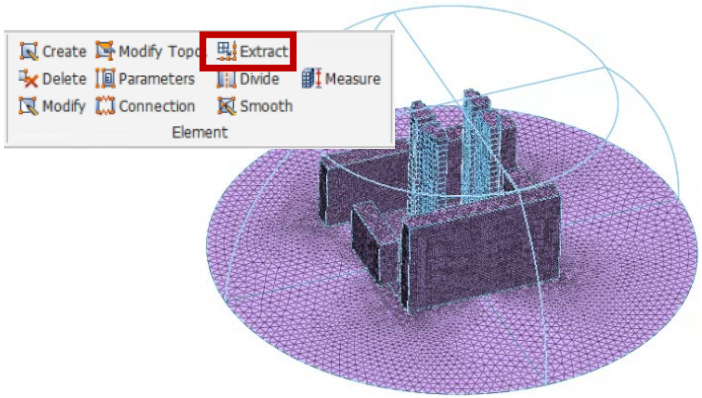


輪廓圖

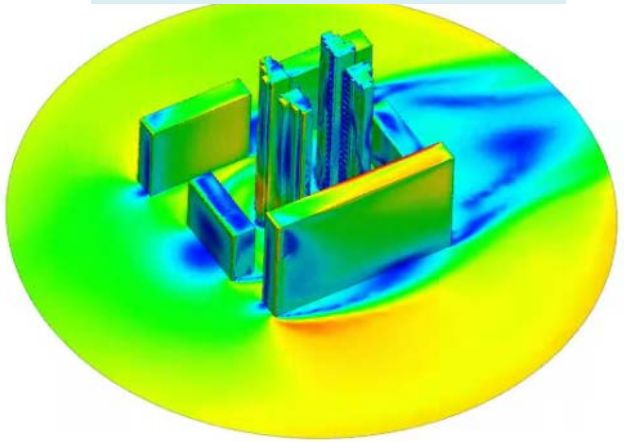


剖面圖

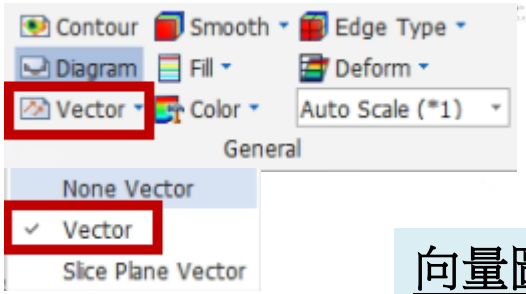
補充:提取流場和建物相接表面元素



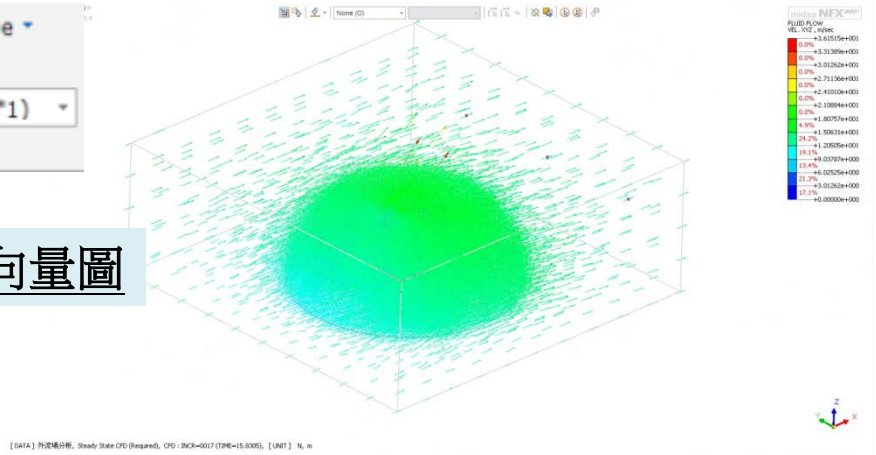
地表和建物表面風速

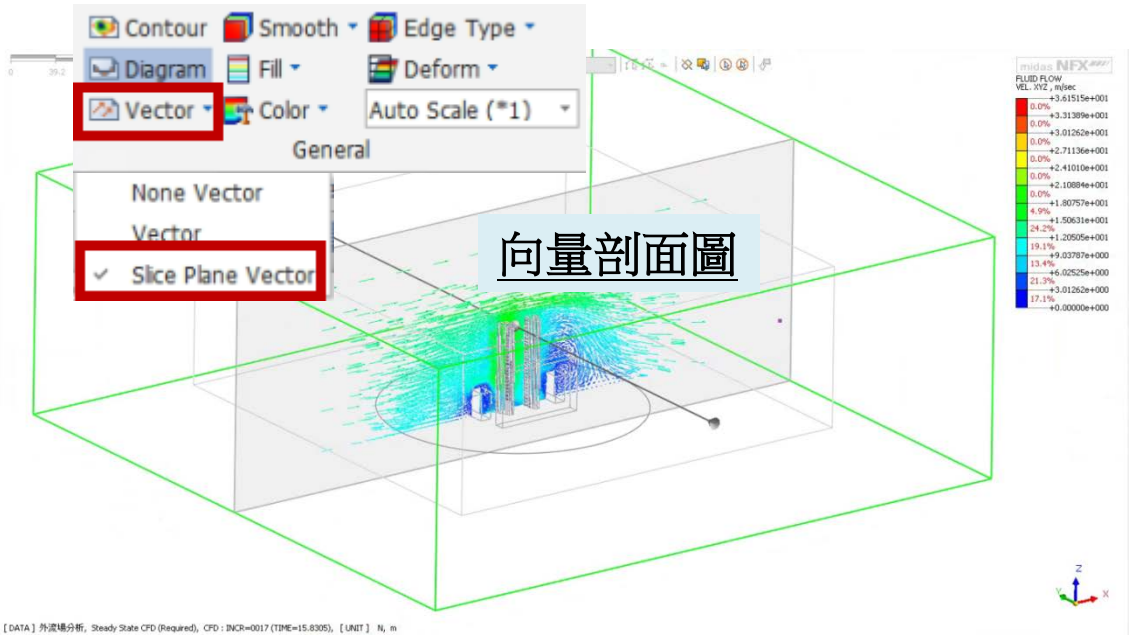


分析結果-2



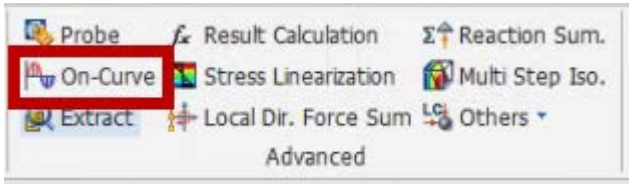
向量圖

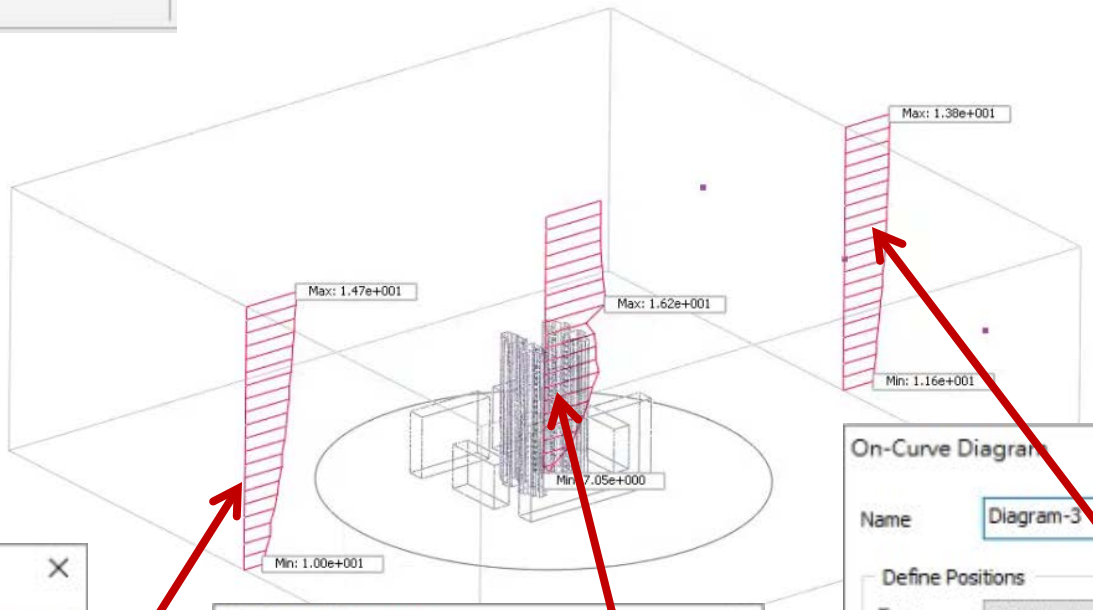




向量剖面圖

分析結果-3





On-Curve Diagram

Name: Diagram-1

Define Positions

Type: 2-Points Line

Point 1: -200, 0, 0

Point 2: -200, 0, 150

Direction: (+)X Dir. Division: 20

OK Cancel Apply

On-Curve Diagram

Name: Diagram-2

Define Positions

Type: 2-Points Line

Point 1: 200, 0, 0

Point 2: 200, 0, 150

Direction: (+)X Dir. Division: 20

OK Cancel Apply

On-Curve Diagram

Name: Diagram-3

Define Positions

Type: 2-Points Line

Point 1: 0, 0, 0

Point 2: 0, 0, 150

Direction: (+)X Dir. Division: 20

OK Cancel Apply

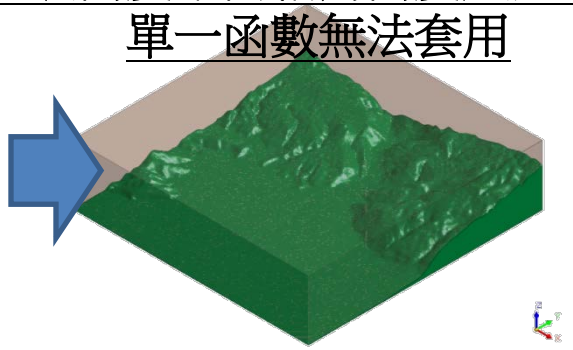
CFD_標準教學系列

強制對流分析103-3D地形風場分析

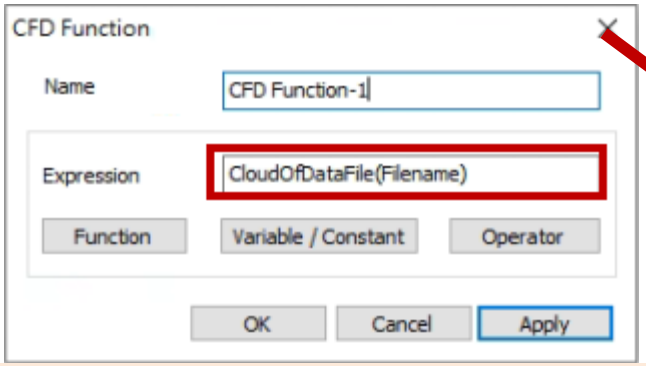
台灣邁達斯

空間座標

地表高度不同相對高度風速不同
單一函數無法套用

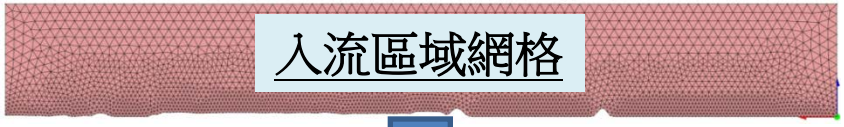


CFD Function讀取空間座標數據
CloudOfDataFile(*.dat)



NFX CFD會依照空間座標內插數值條件
(空間座標間隔過大,數值可能不連續)

入流區域網格



入流區域空間特徵

空間特徵座標流速計算

Inlet(m/sec)

V	10	遞增係數	0.14

相對高度風速選擇

IF(H<10,Vx,IF(H>=10,Vx*(H/10)^0.143))

H-相對地表高度	VX=地表風速

Line1

X (m)	Y (m)	Z (m)	H-相對地表高度	Velocity(m/sec)
5064.72	8396.78	10.19	0.00	10.00
5064.72	839			
5064.72	839			
5064.72	839			
5064.72	839			

Line2

X (m)	Y (m)	Z (m)	H-相對地表高度	Velocity(m/sec)
4800.00	8396.78	4.12	0.00	10.00
4800.00	8396.78	173.90	169.78	15.73
4800.00	8396.78	343.68	339.56	17.58
4800.00	8396.78	513.46	509.34	18.76
4800.00	8396.78	683.24	679.12	19.64

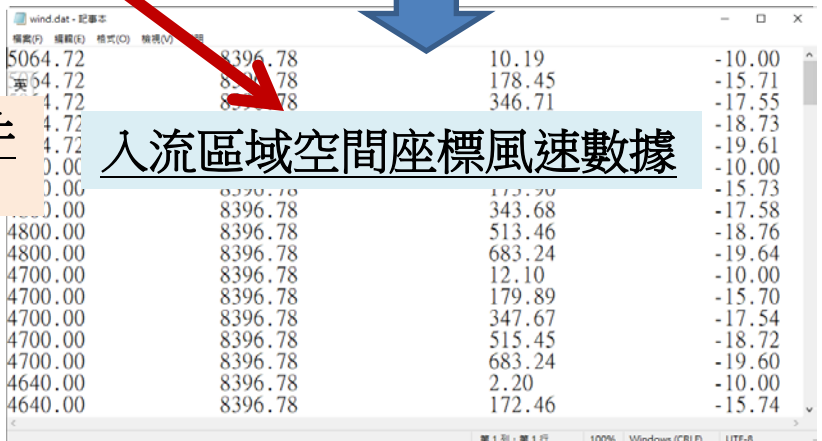
Line3

X (m)	Y (m)	Z (m)	H-相對地表高度	Velocity(m/sec)
4700.00	8396.78	12.10	0.00	10.00
4700.00	8396.78	179.89	167.78	15.70
4700.00	8396.78	347.67	335.57	17.54
4700.00	8396.78	515.45	503.35	18.72
4700.00	8396.78	683.24	671.13	19.60

Line4

X (m)	Y (m)	Z (m)	H-相對地表高度	Velocity(m/sec)
4640.00	8396.78	2.20	0.00	10.00
4640.00	8396.78	172.46	170.26	15.74

X (m)	Y (m)	Z (m)	Velocity(m/sec)
5064.72	8396.78	10.19	-10.00
5064.72	8396.78	178.45	-15.71
5064.72	8396.78	346.71	-17.55
5064.72	8396.78	514.97	-18.73
5064.72	8396.78	683.24	-19.61
4800.00	8396.78	4.12	-10.00
4800.00	8396.78	173.90	-15.73
4800.00	8396.78	343.68	-17.58
4800.00	8396.78	513.46	-18.76
4800.00	8396.78	683.24	-19.64
	6.78	12.10	-10.00
	6.78	179.89	-15.70
	6.78	347.67	-17.54
	6.78	515.45	-18.72
	6.78	683.24	-19.60
	6.78	2.20	-10.00
4640.00	8396.78	172.46	-15.74
4640.00	8396.78	342.72	-17.58
4640.00	8396.78	512.98	-18.76
4640.00	8396.78	683.24	-19.65
4540.00	8396.78	6.78	-10.00
4540.00	8396.78	175.89	-15.72
4540.00	8396.78	345.01	-17.57
4540.00	8396.78	514.12	-18.74
4540.00	8396.78	683.24	-19.63
4480.00	8396.78	22.27	-10.00
4480.00	8396.78	187.51	-15.66
4480.00	8396.78	352.75	-17.50
4480.00	8396.78	518.00	-18.67
4480.00	8396.78	683.24	-19.55
4420.00	8396.78	4.20	-10.00
4420.00	8396.78	173.96	-15.73
4420.00	8396.78	343.72	-17.58
4420.00	8396.78	513.48	-18.75
4420.00	8396.78	683.24	-19.64



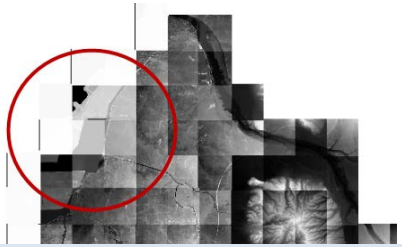
入流區域空間座標風速數據

midas NFX-CFD

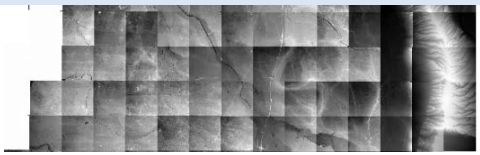
彰濱工業區DEM地形圖

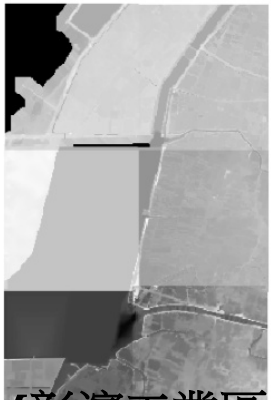
Google Map

(QGIS軟體操作)



內政部20公尺網格數值地形模型資料

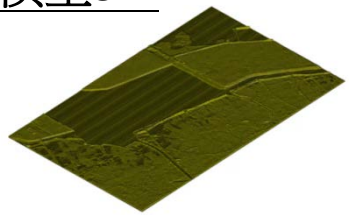




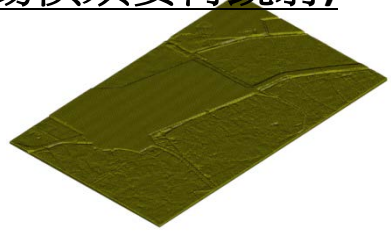
DEM彰濱工業區週邊地形模型資料

DEM合併地形模型

地形模型STL



流場模型STL
(流場模須要再鏡射)



Reference

內政部20公尺網格數值地形模型資料 https://data.gov.tw/dataset/35430

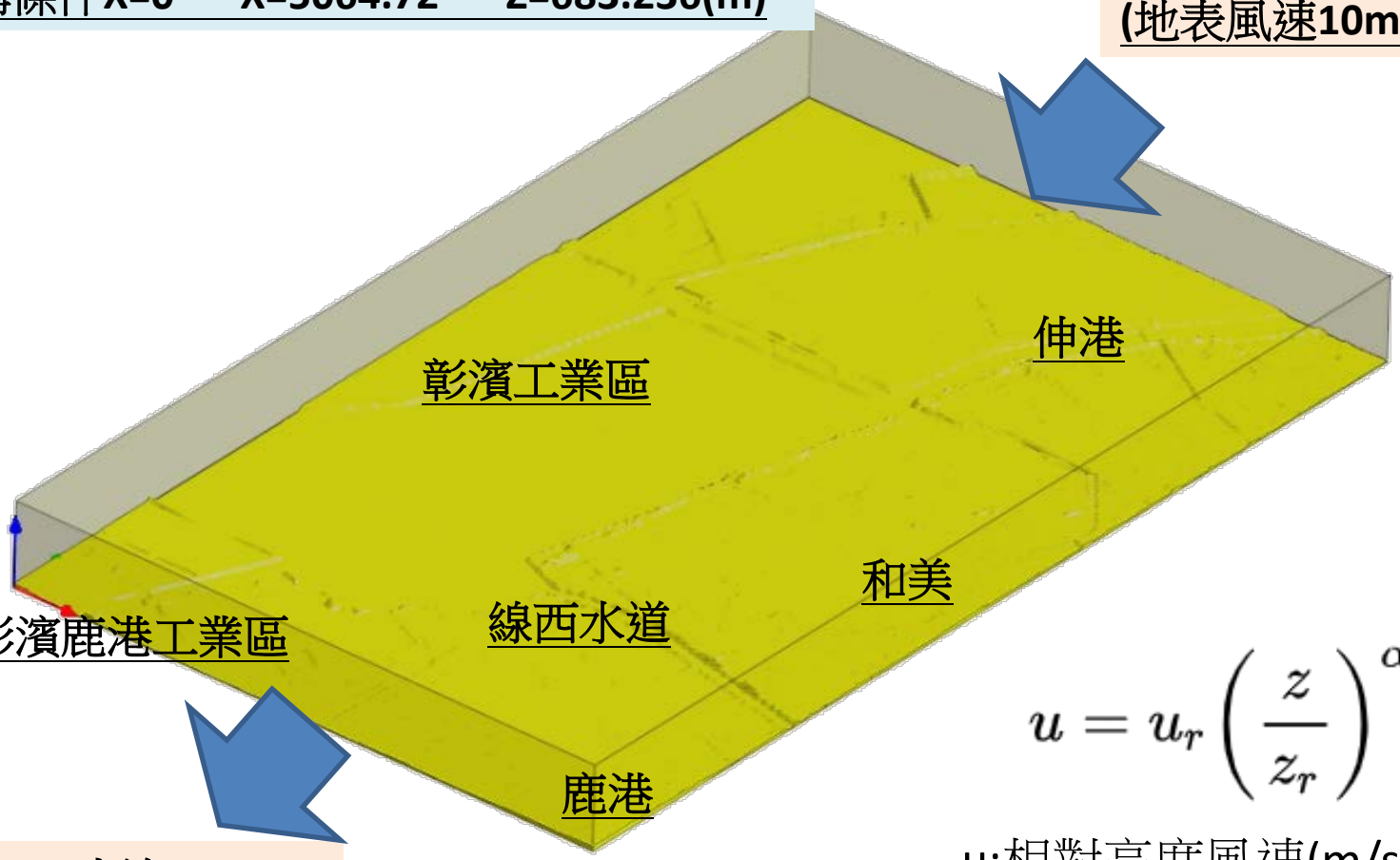
QGIS軟體 https://qgis.org/zh-Hant/site/forusers/download.html

MIDAS

分析說明

對稱條件X=0 、 X=5064.72 、 Z=683.236(m)

入流口
(地表風速10m/s)

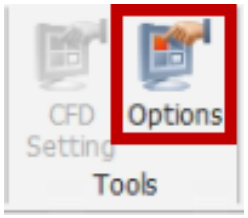


出流口
(Pressure=0N/m²)

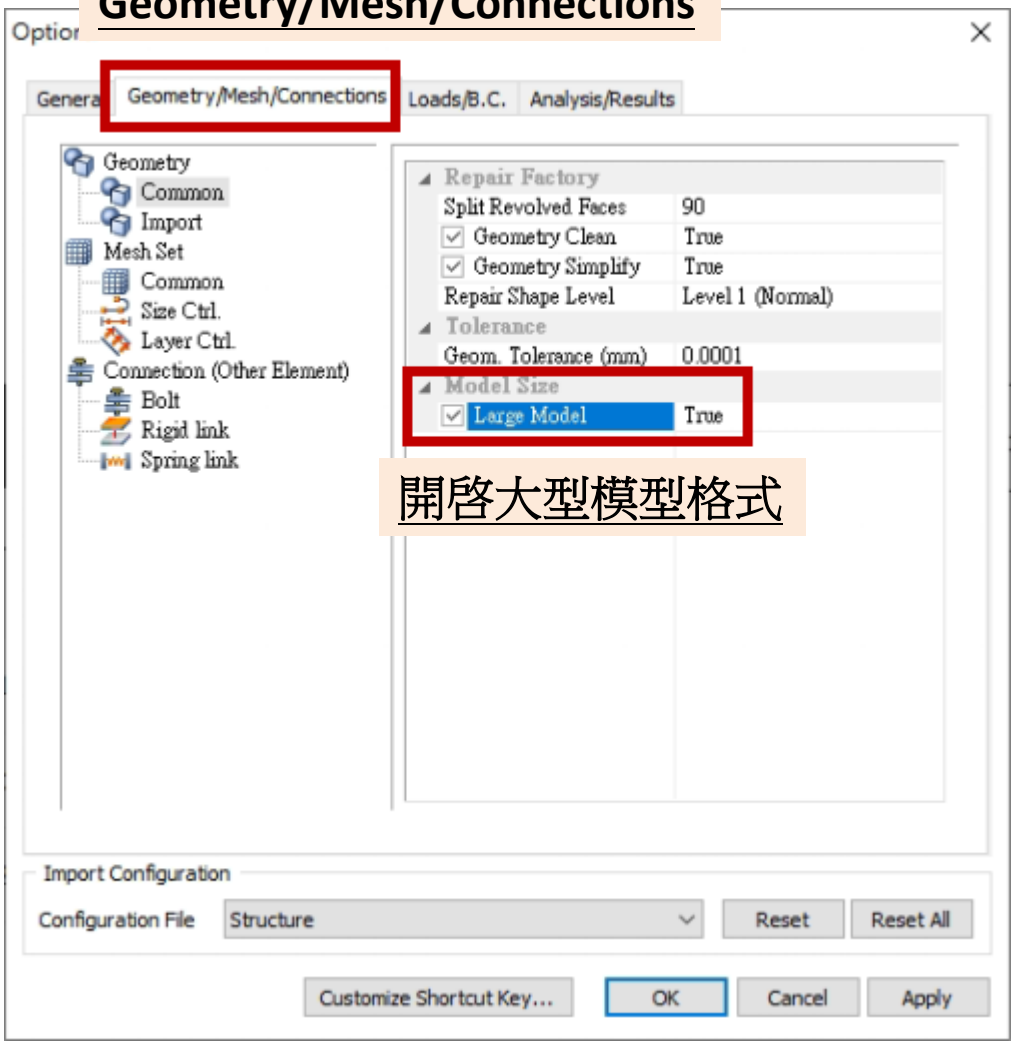
$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

u:相對高度風速(m/s)
Z_γ:地表高度(10m)
u_γ:地表高度Z_γ風速(假設10m/s)
α :0.143

大型模型格式



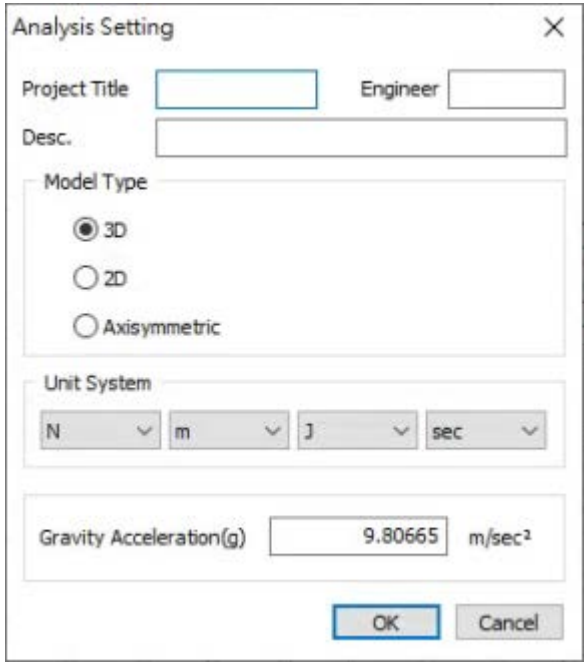
Geometry/Mesh/Connections



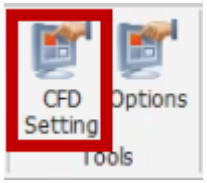
開啓大型模型格式

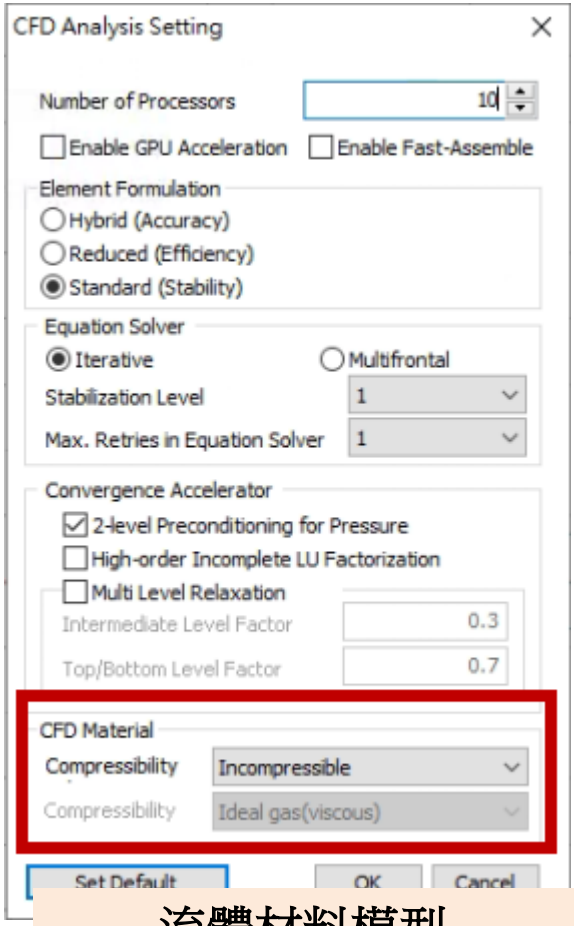
註:大型模型格式,匯出幾何格式會自動調整尺寸比例1/1000。

環境



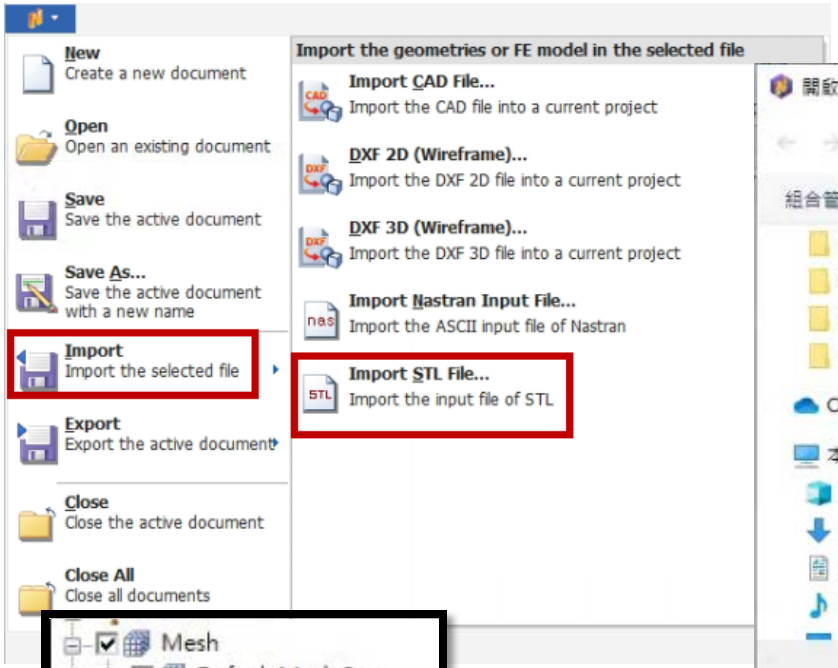
單位使用N/m/J/sec



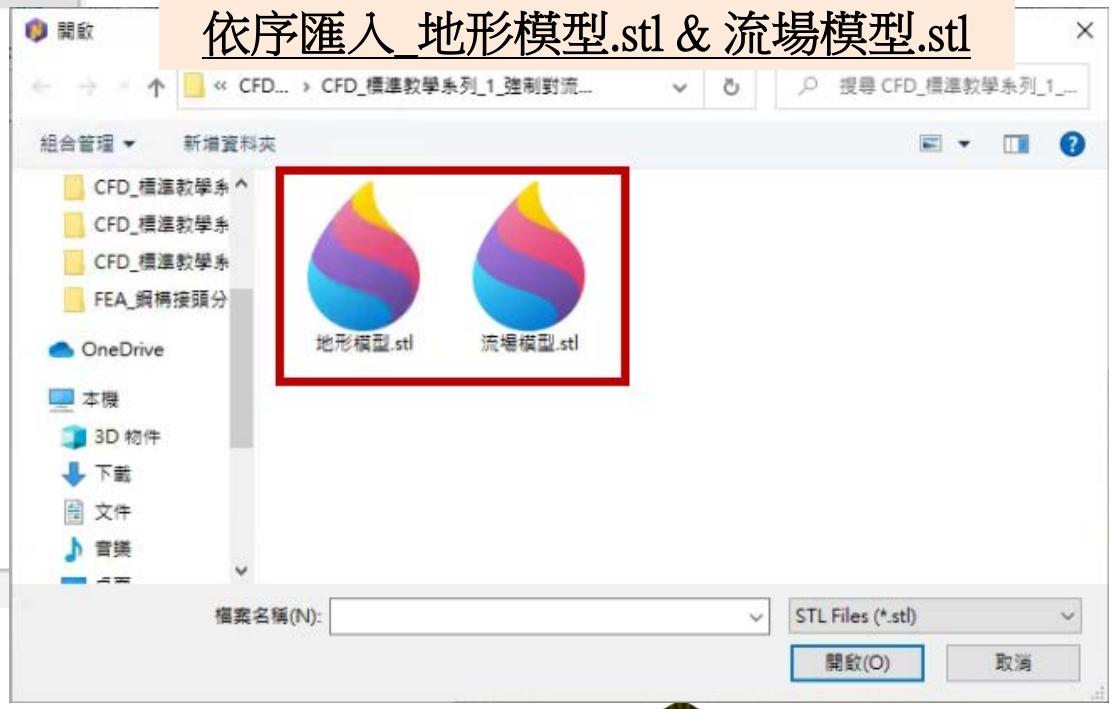


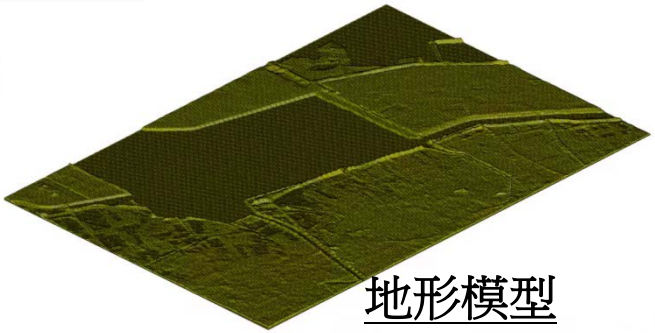
流體材料模型
不可壓縮流

匯入STL模型

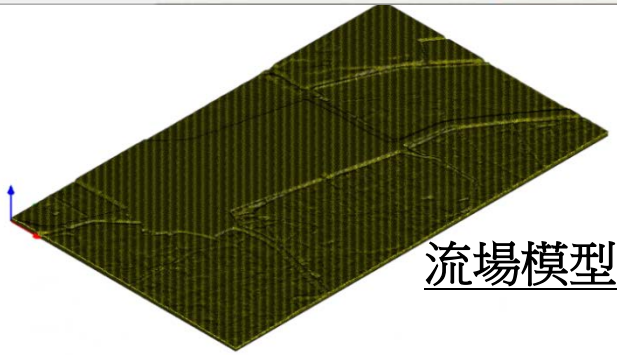


依序匯入_地形模型.stl & 流場模型.stl





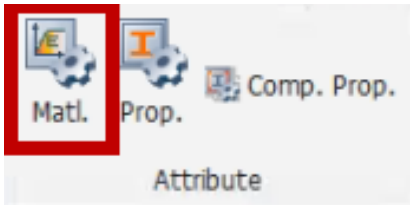
地形模型

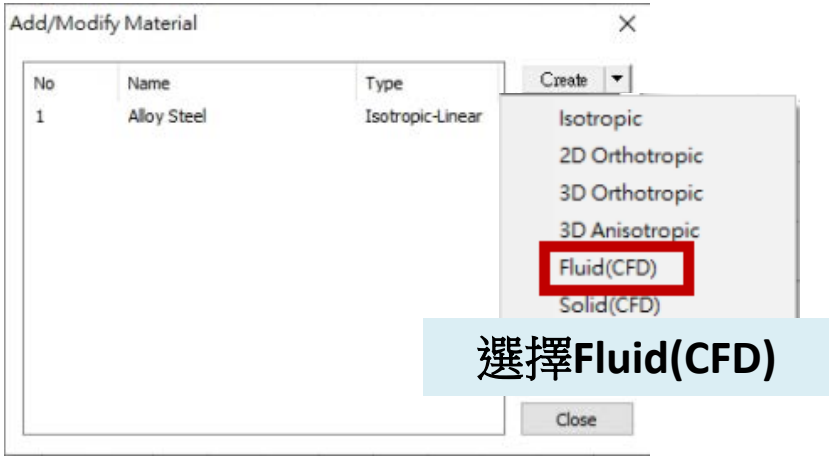


流場模型

註1.QGIS軟體匯出之流場模型必須做空間鏡射
註2:STL匯入只會有外表面元素特徵(2D Shell)

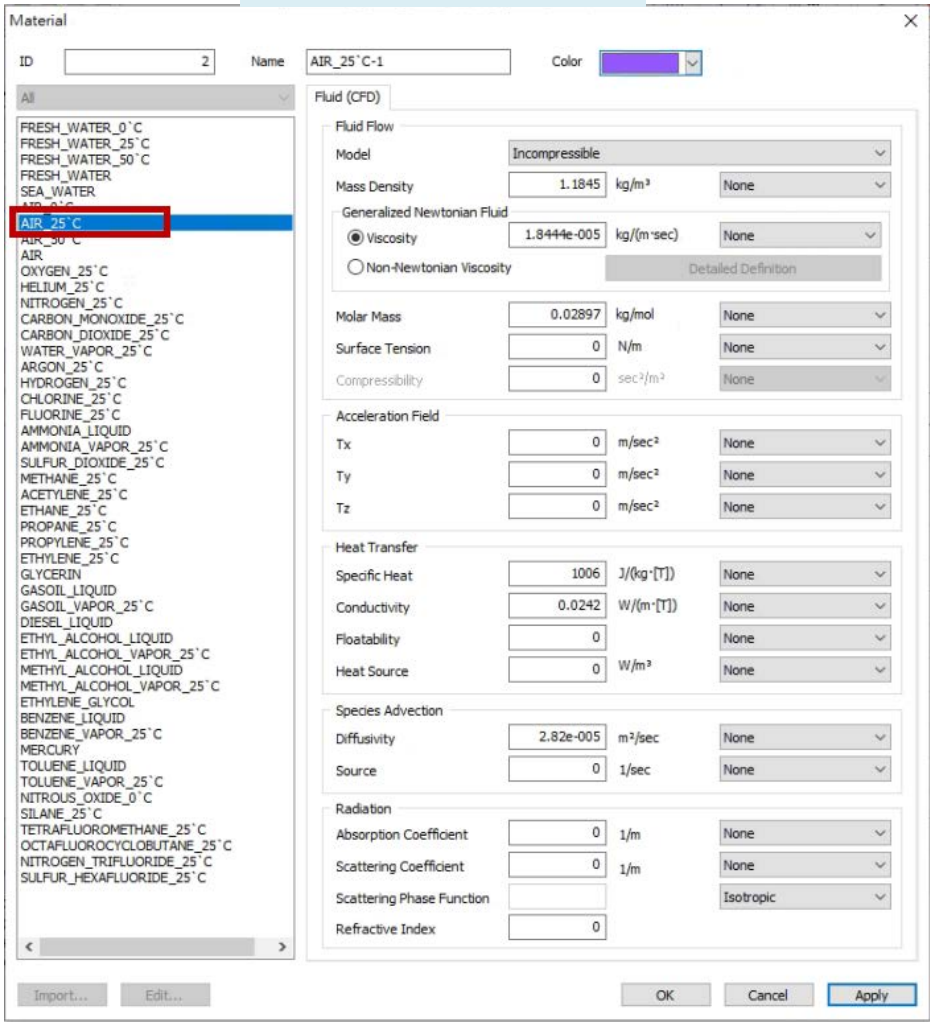
材料



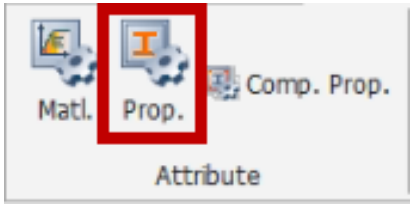


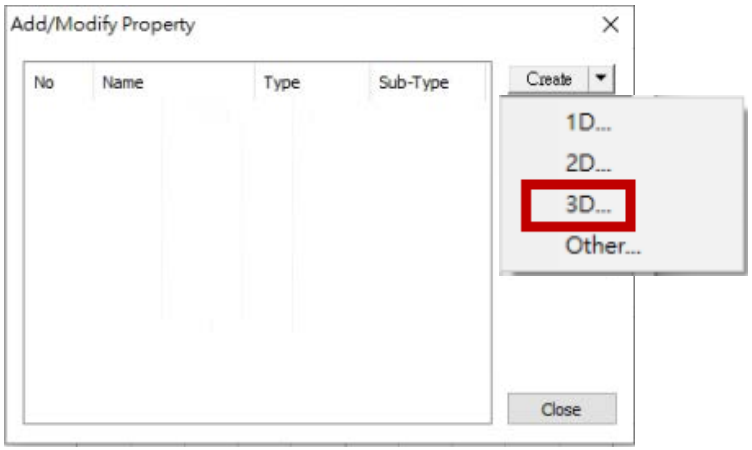
選擇Fluid(CFD)

新增AIR 25 ℃

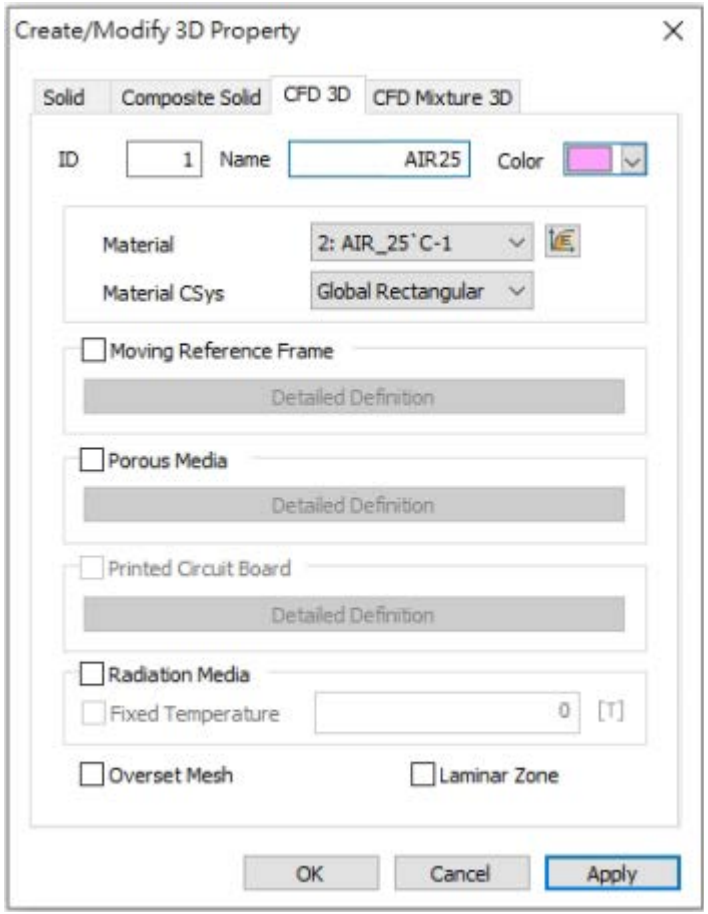


屬性



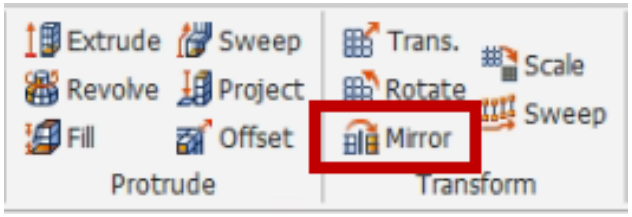


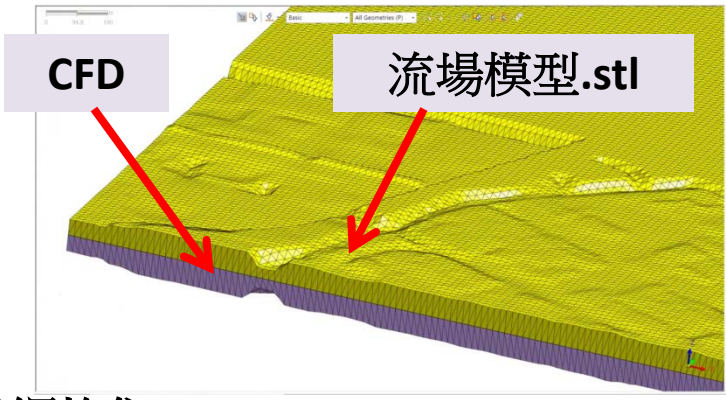
新增AIR 25 ℃ Property



midas NFX-CFD

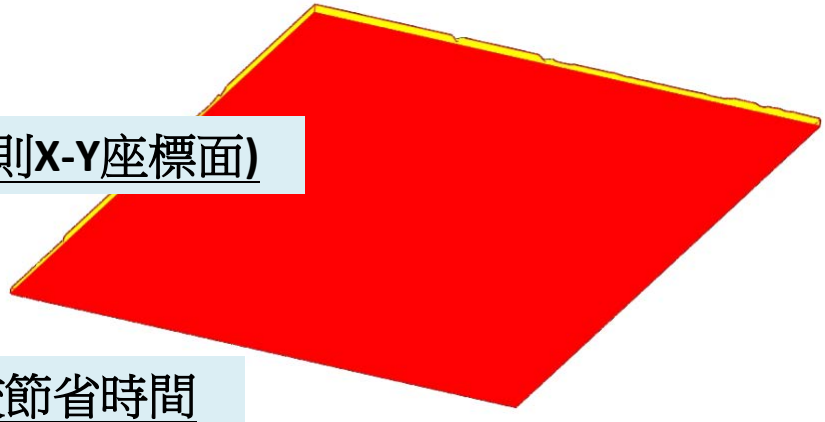
流場域-鏡射





鏡射流場域網格集

輸入對稱面座標(or選則X-Y座標面)



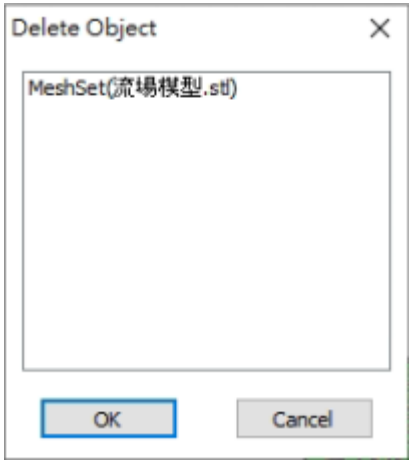
勾選Copy Object較節省時間

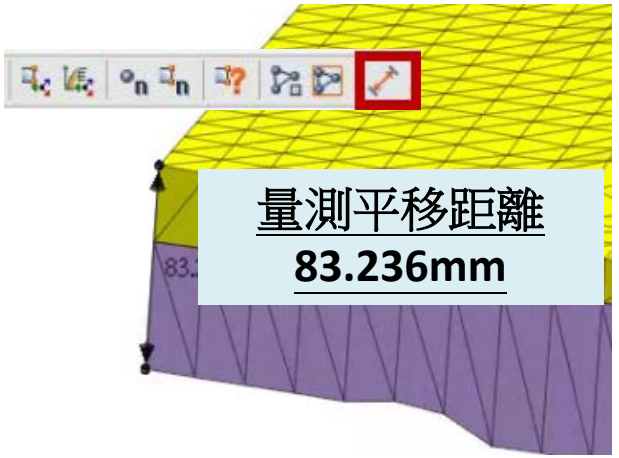
自訂網格集名稱CFD

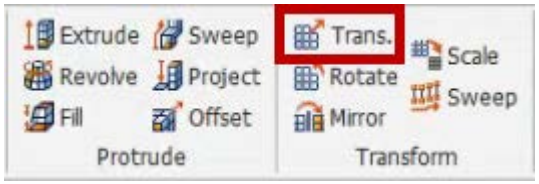
刪除&平移網格集

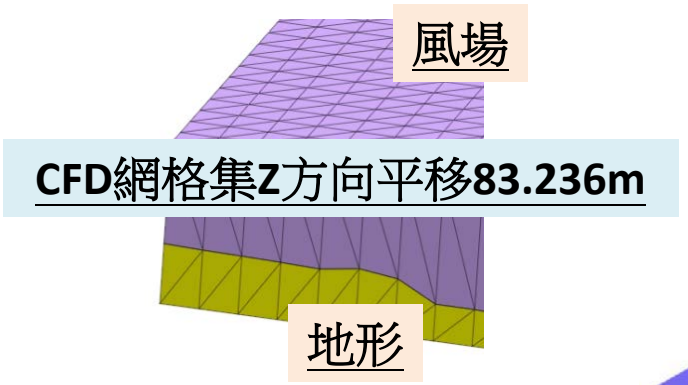
(刪除初始流場域/平移CFD網格集)

鍵盤Delete



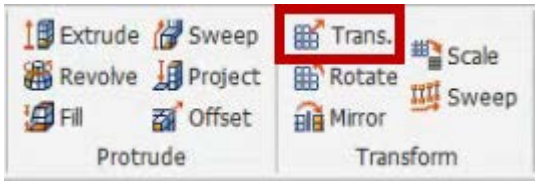


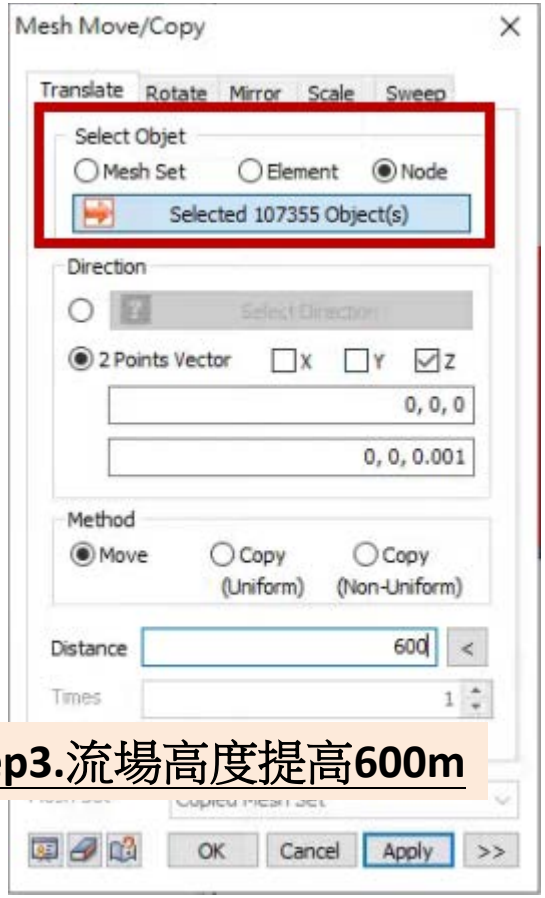




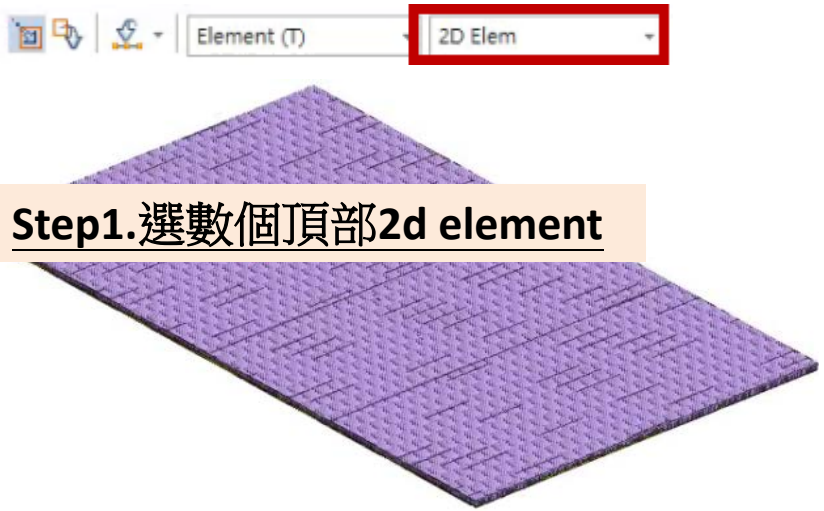
平移網格集

(提高風場模型節點高度)





Step3.流場高度提高600m

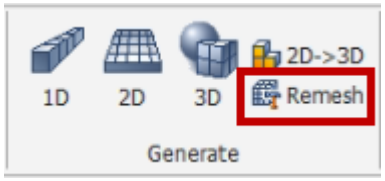


Step1.選數個頂部2d element

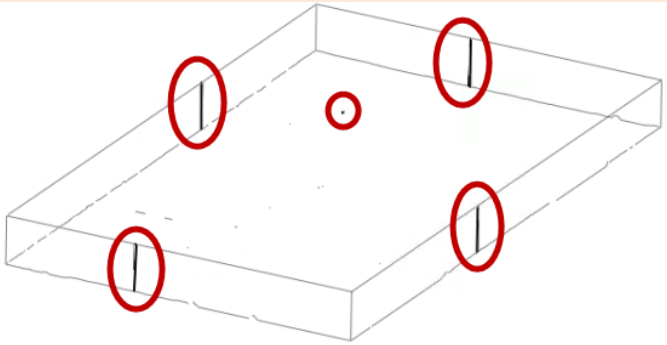
Step2.特徵角選取
(自動全選頂部所有元素之節點)

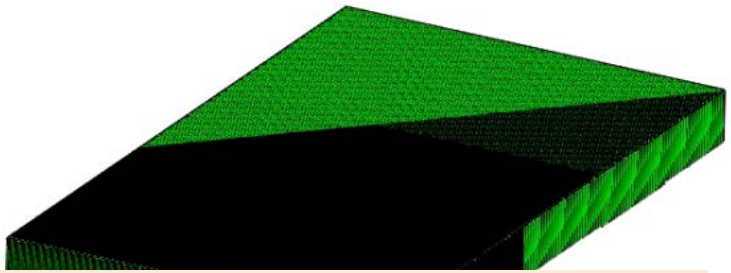
重新2D網格劃分

(風場模型四周和頂部)

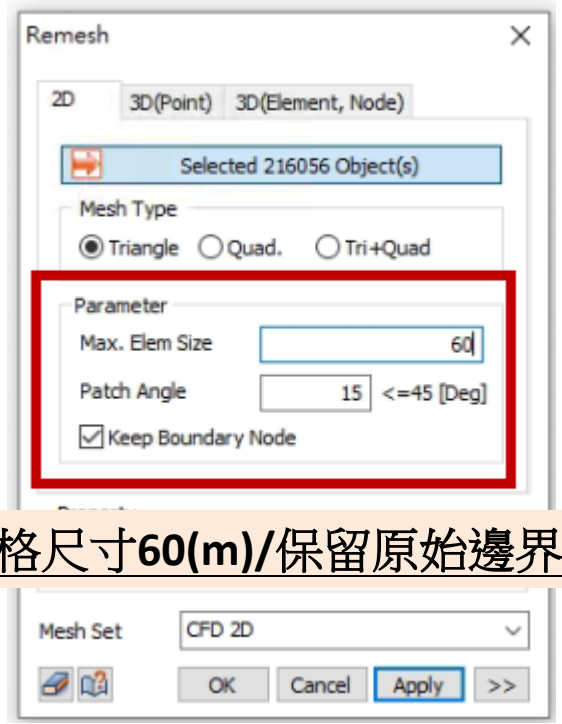


Step1.選數個頂部和四周2d element





Step2.特徵角選取
(自動全選頂部和四周所有元素)



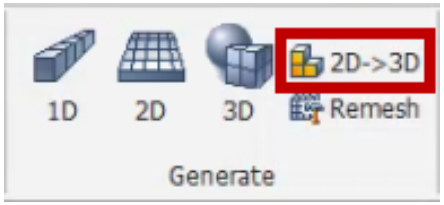
Step3.網格尺寸60(m)/保留原始邊界節點

Step4.自訂網格集名稱CFD_2D

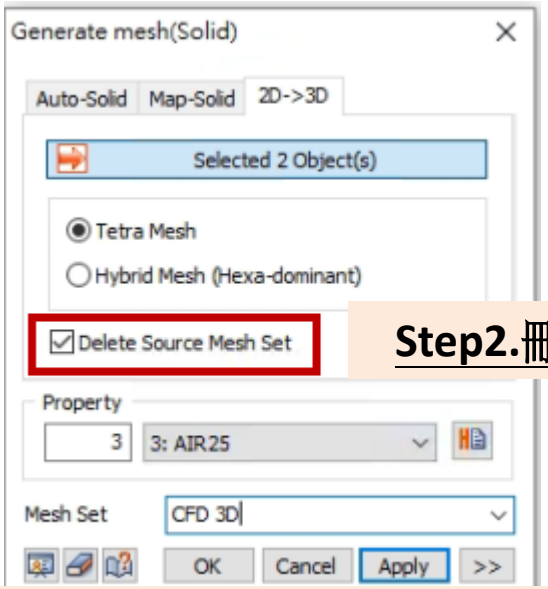
midas NFX-CFD

3D網格

(封閉2D元素生成3D Solid元素)

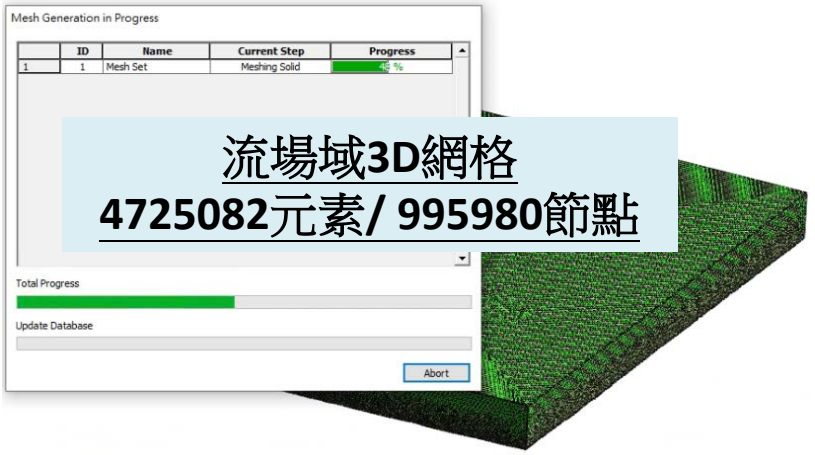


Step1.選取流場域封閉2D元素網格集



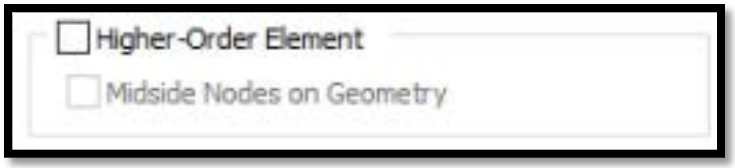
Step2.刪除封閉2D元素網格集

Step3.自訂網格集名稱CFD_3D



流場域3D網格
4725082元素/ 995980節點

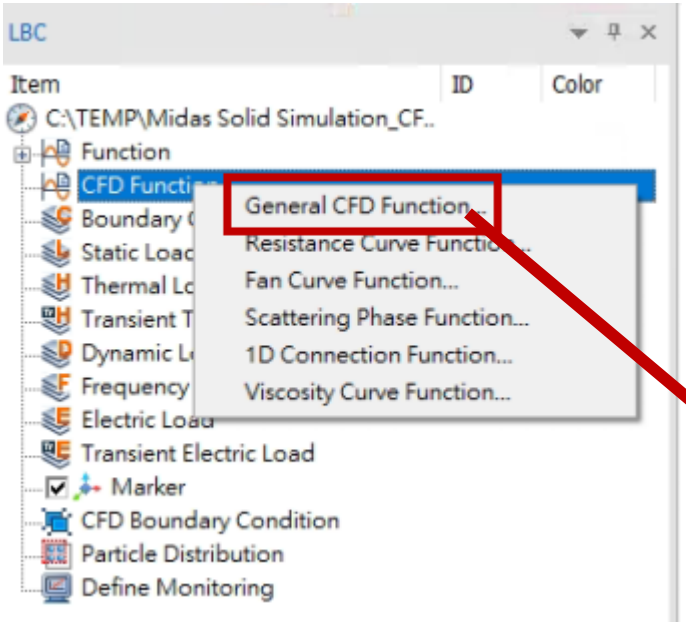
NFX CFD不支援二階(高階)元素

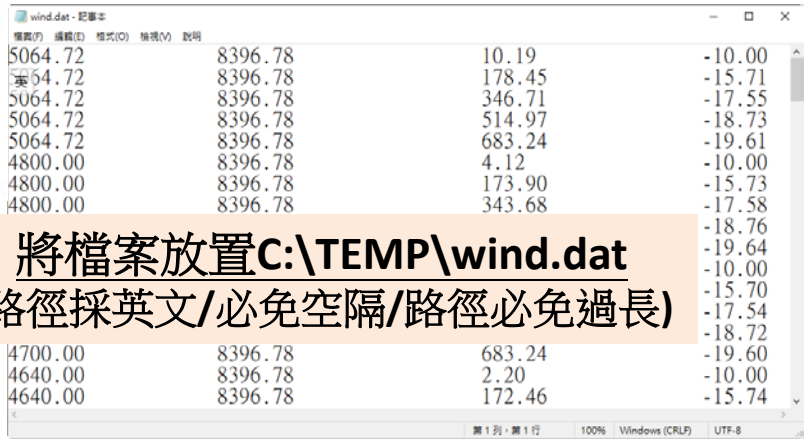


MIDAS

入流-1

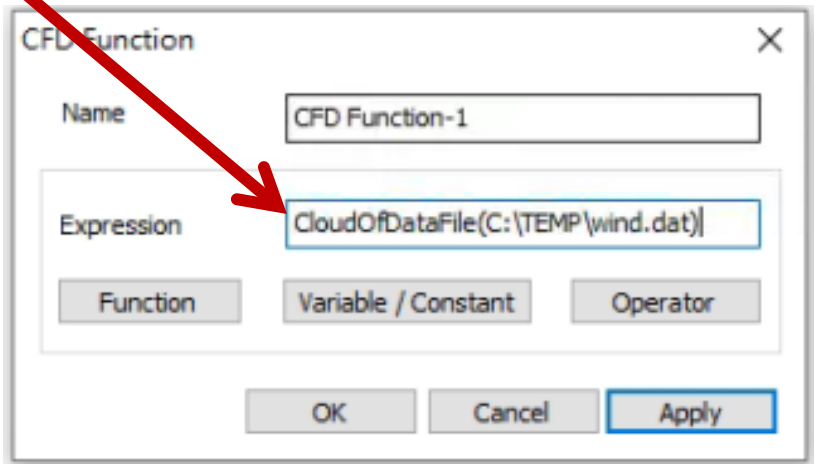
(地形高度指定相對高度風速函數)





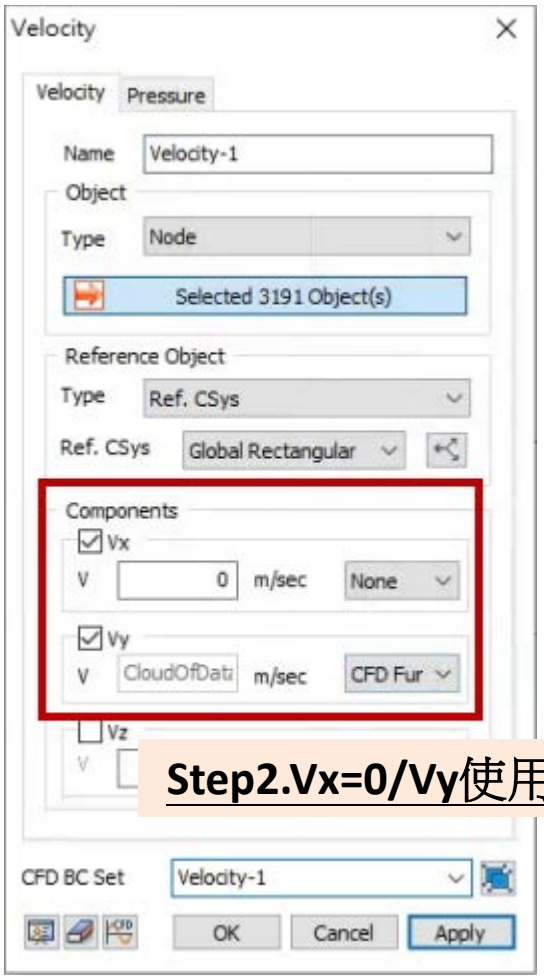
將檔案放置C:\TEMP\wind.dat

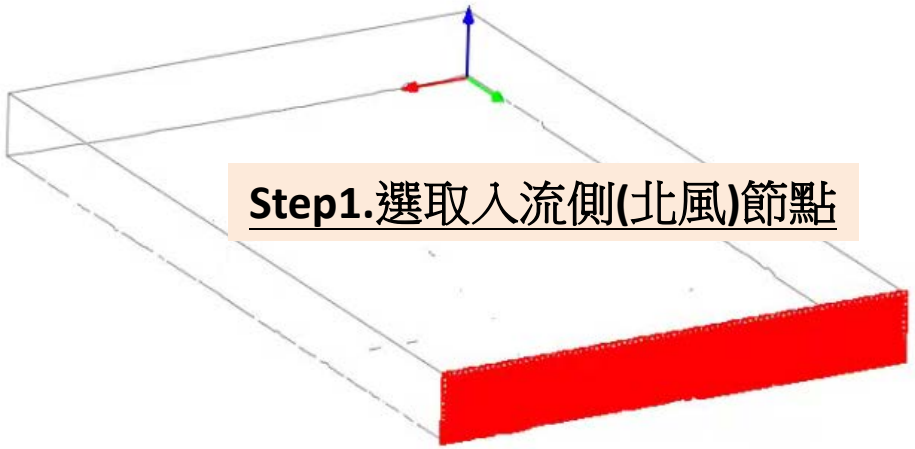
(路徑採英文/必免空隔/路徑必免過長)



入流-2

右側視圖框選



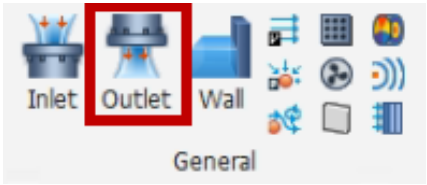


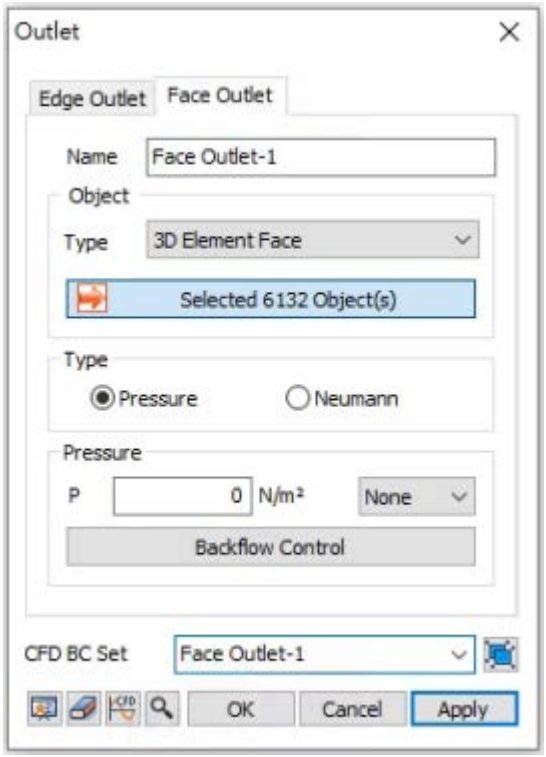
Step1.選取入流側(北風)節點

Step2.Vx=0/Vy使用函數

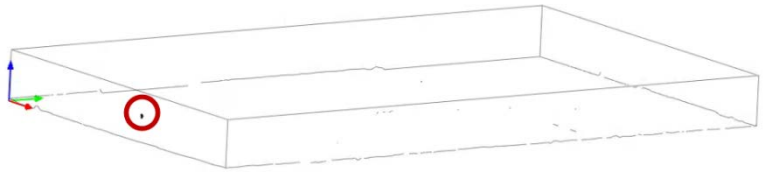
註:不同邊界&載荷建議使用不同集合名稱。

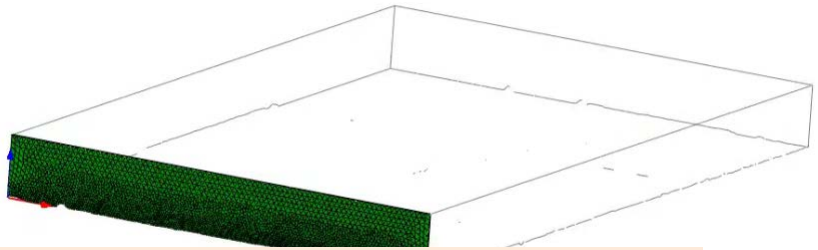
出流





Step1出流側數個元素

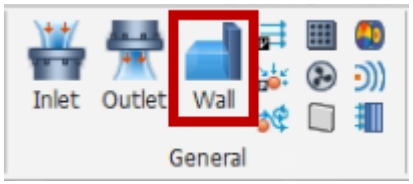




Step2.特徵角選取
(自動全選出流側所有元素)

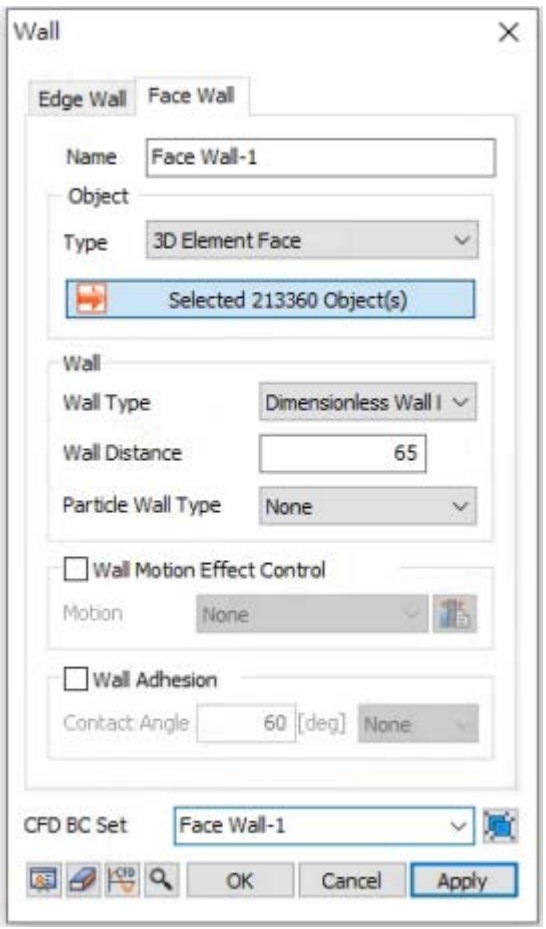
壁函數-地面和流場交界

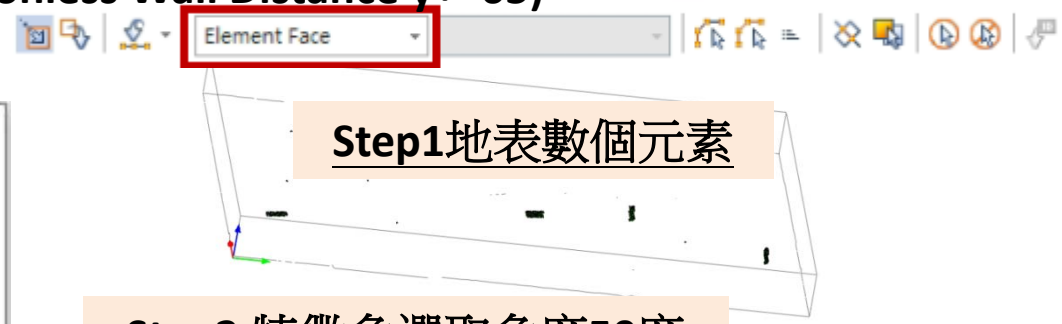
(Wall)



壁函數施加區域-流場和地表交界特徵

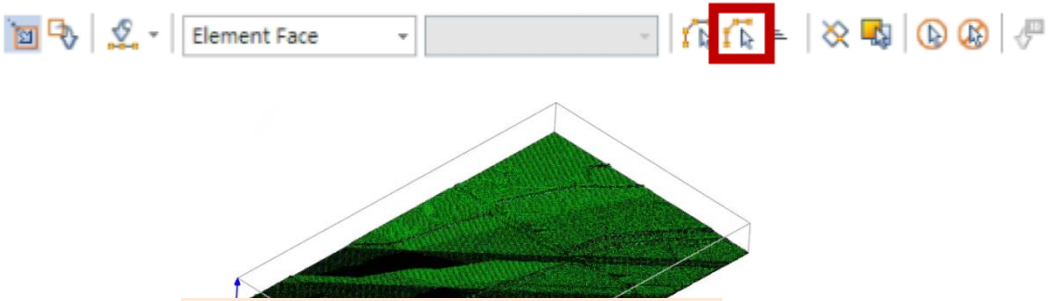
(Dimensionless Wall Distance y+=65)





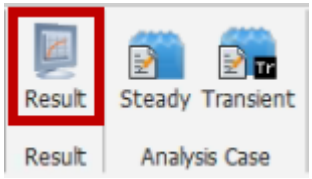
Step1地表數個元素

Step2.特徵角選取角度50度

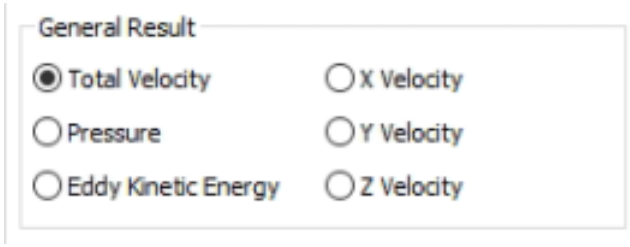


Step3.特徵角選取
(自動選出地表所有元素)

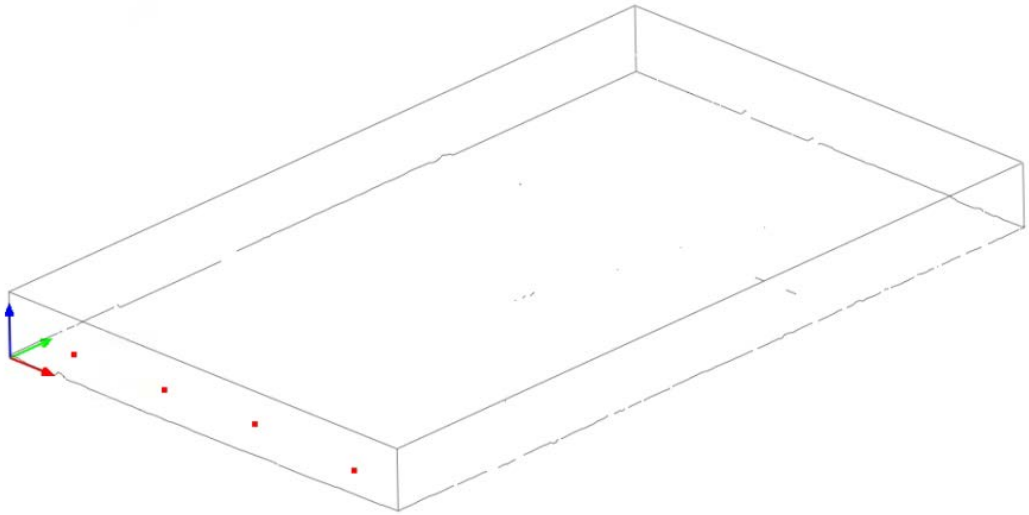
觀測點



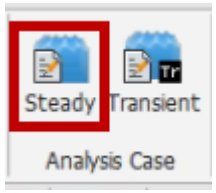
輸出特定節點(速度)數據



選取出流口節點作為觀測節點

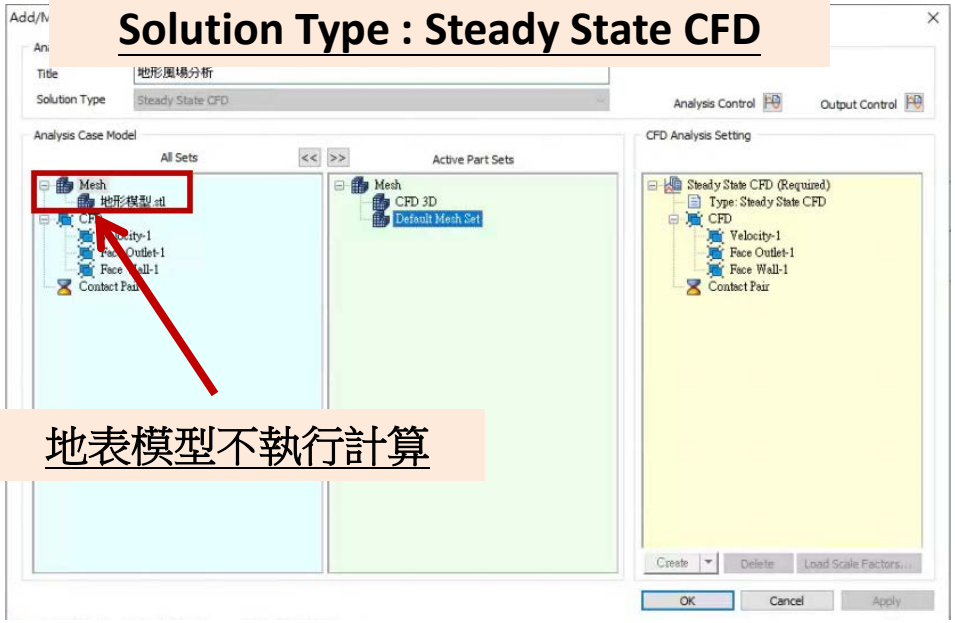


CFD穩態分析-1

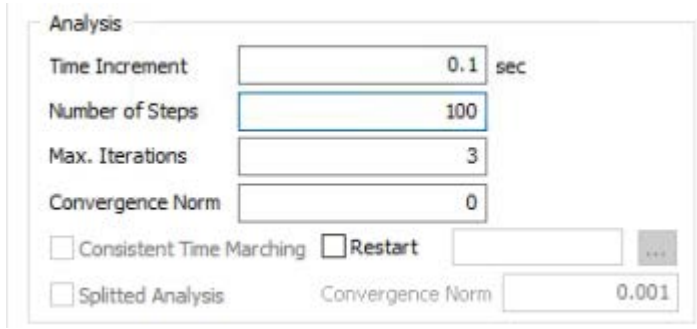


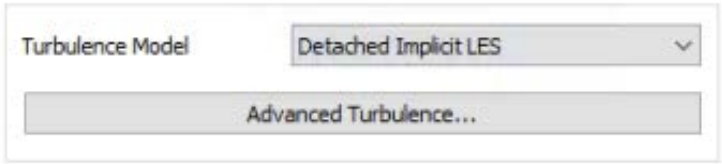
範例計算量大,自行判斷增量數量和收斂條件

Solution Type : Steady State CFD

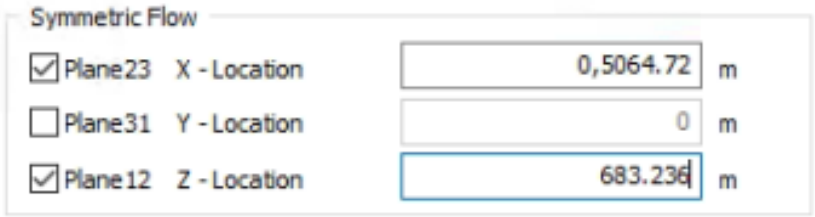


地表模型不執行計算

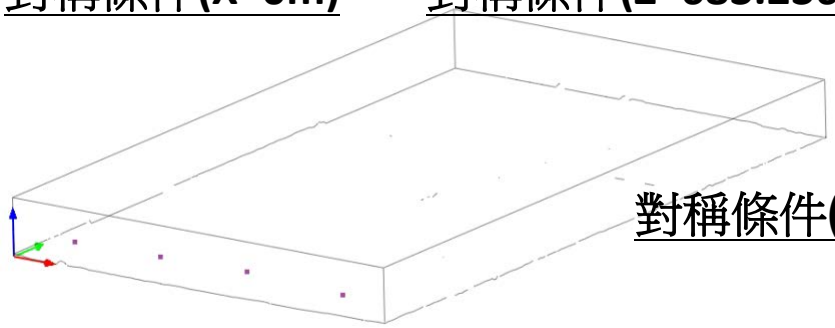




CFD穩態分析-2
(對稱條件)

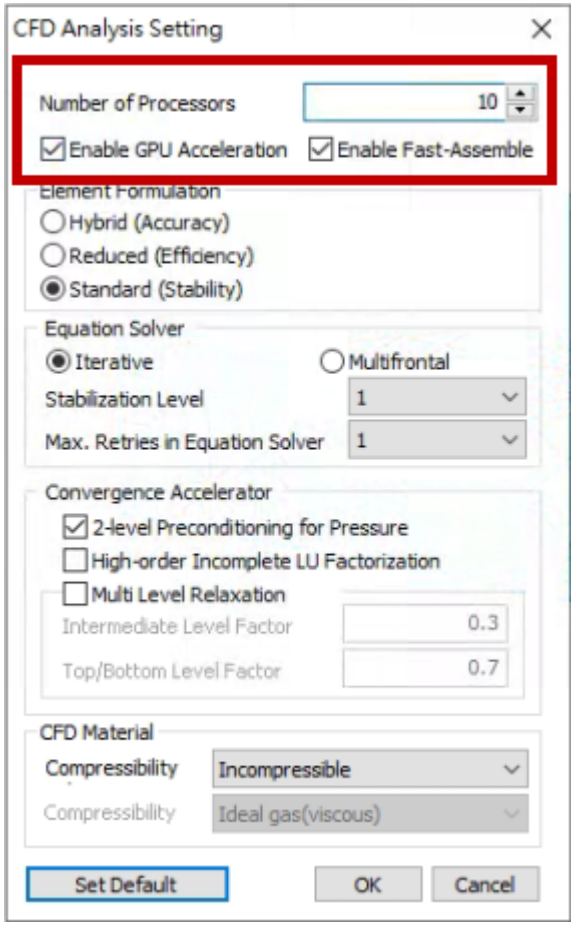


對稱條件(X=0m)　　對稱條件(Z=683.236m)



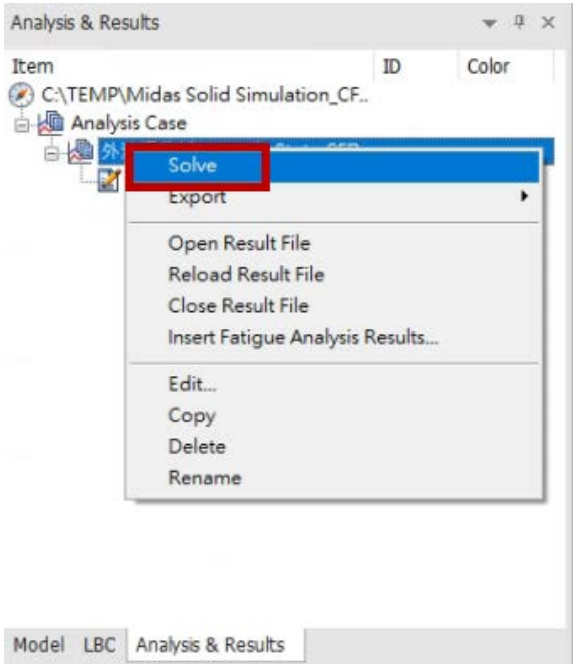
對稱條件(X=5064.72m)

CPU&GPU定義


CFD Analysis Setting
Number of Processors
10
Enable GPU Acceleration
Enable Fast-Assemble
Element Formulation
Hybrid (Accuracy)
Reduced (Efficiency)
Standard (Stability)
Equation Solver
Iterative
Multifrontal
Stabilization Level
1
Max. Retries in Equation Solver
1
Convergence Accelerator
2-level Preconditioning for Pressure
High-order Incomplete LU Factorization
Multi Level Relaxation
Intermediate Level Factor
0.3
Top/Bottom Level Factor
0.7
CFD Material
Compressibility
Incompressible
Compressibility
Ideal gas(viscous)
Set Default
OK
Cancel

執行CFD求解

(範例計算量大,自行判斷增量數量和收斂條件)



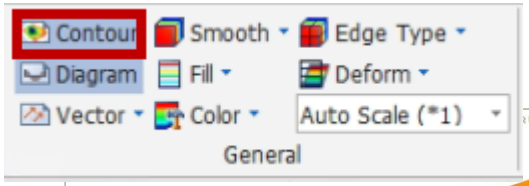
速度/壓力迭代收斂曲線判斷每次迭代步是否收斂

監測點達到穩定和達到收斂可以停止計算

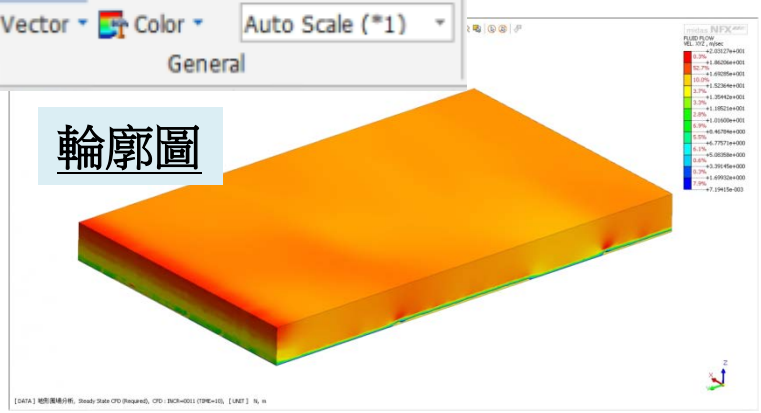
註1:透過收斂曲線和觀測點數據數據判斷是否斂

註2:如果無法收斂和達到穩定,判斷邊界是否合理和加密網格

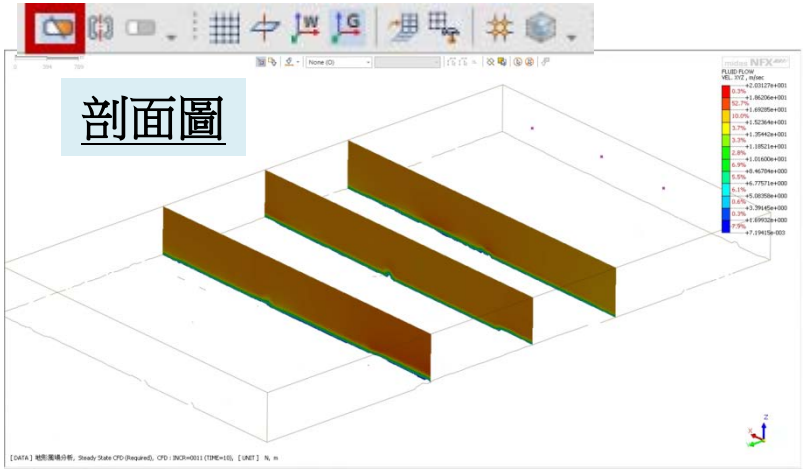
分析結果-1



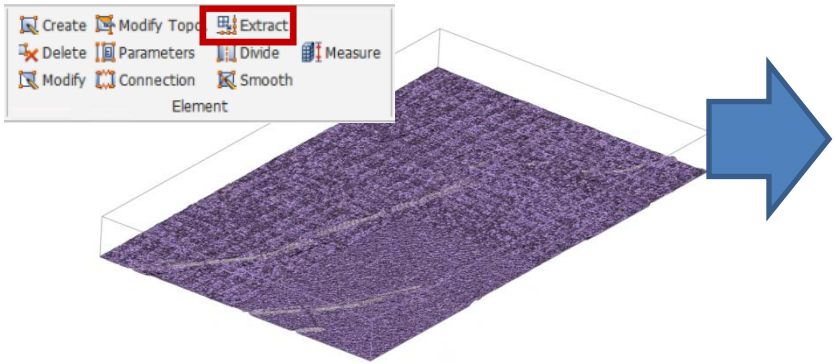
輪廓圖



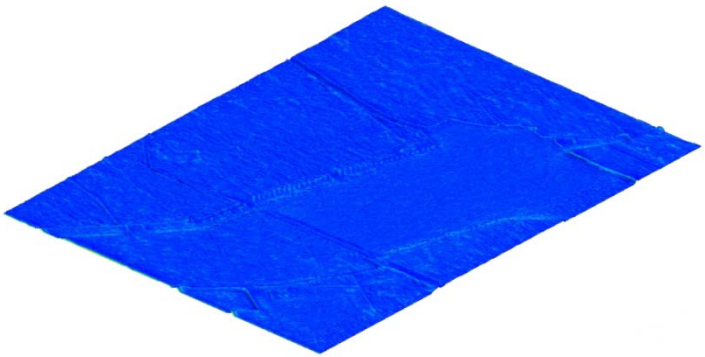
剖面圖



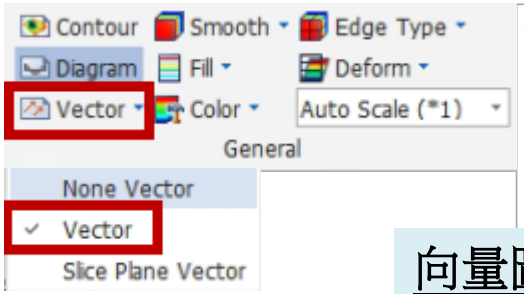
補充:提取地表面元素



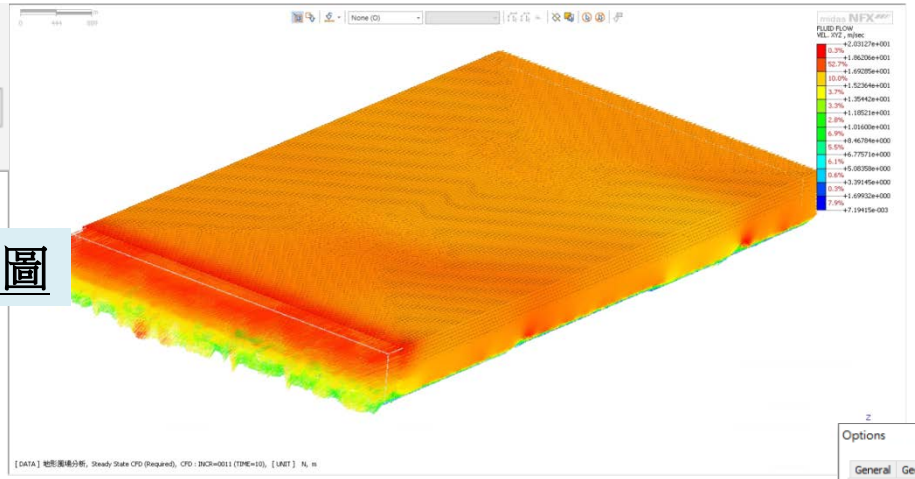
地表面風速



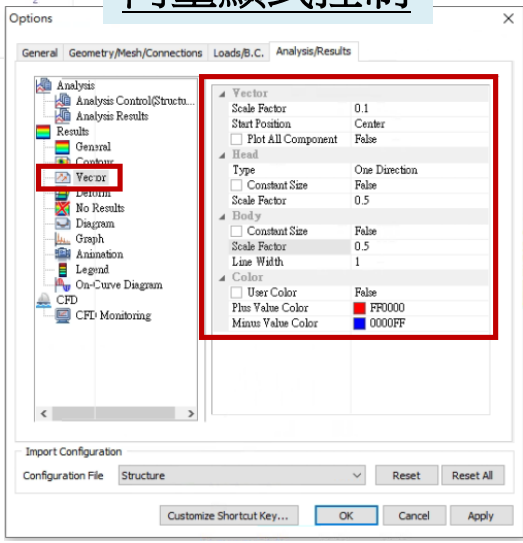
分析結果-2

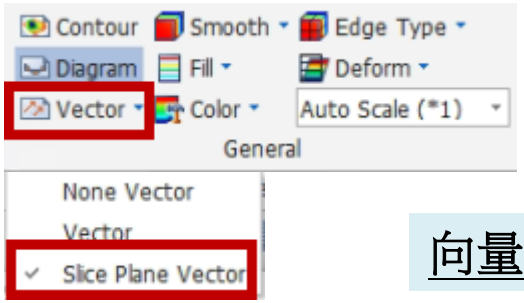


向量圖

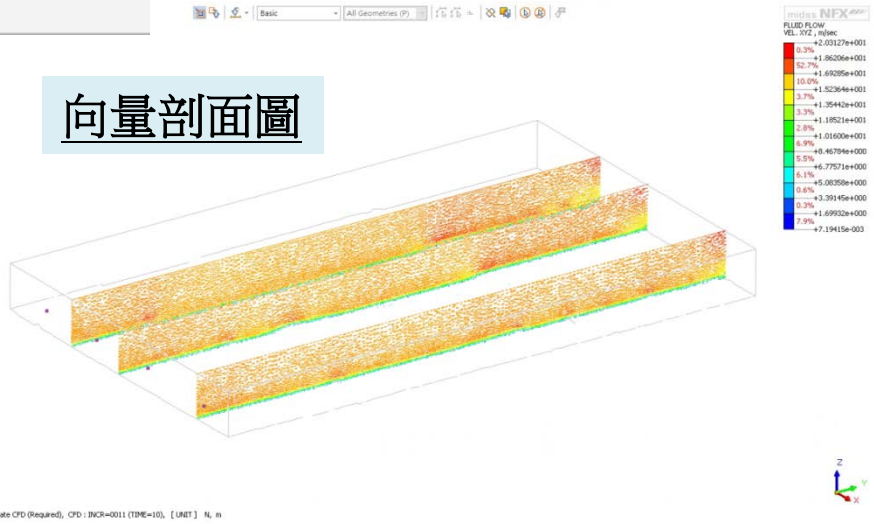


向量顯式控制



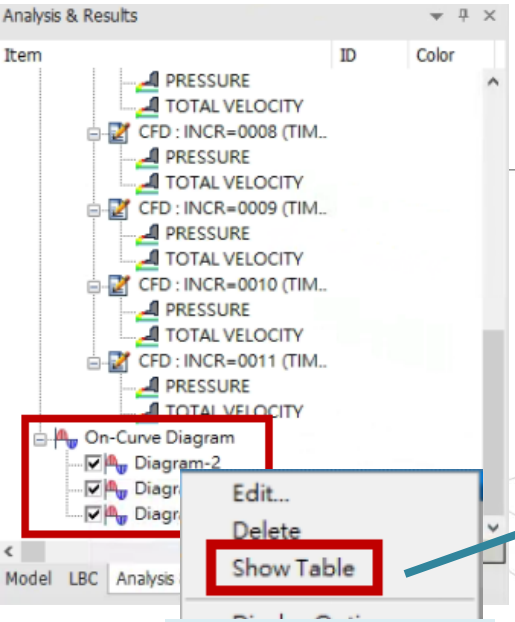


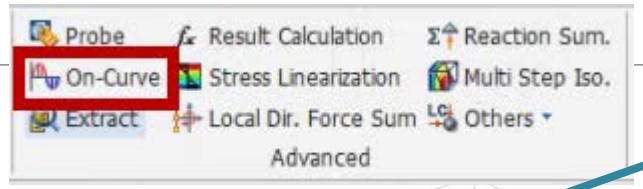
向量剖面圖



midas NFX-CFD

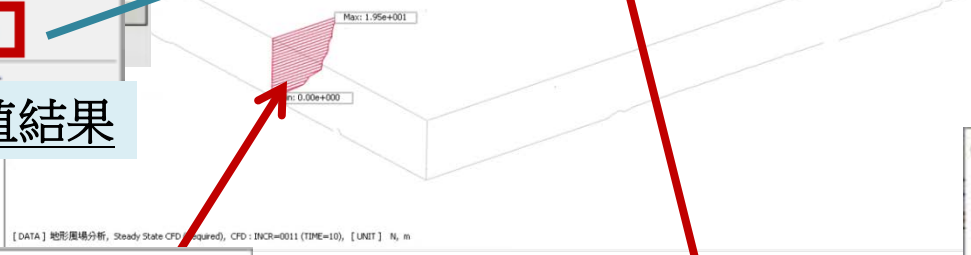
分析結果-3

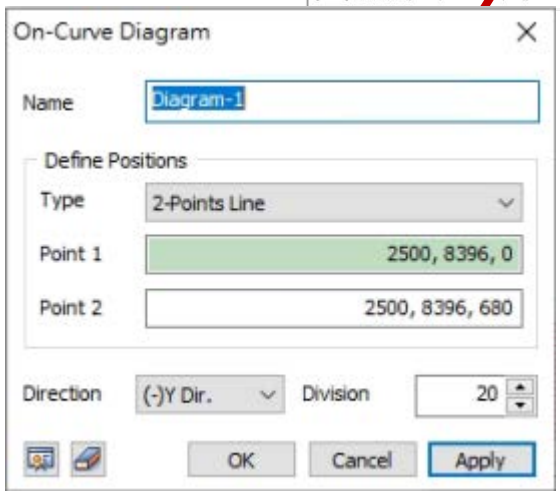


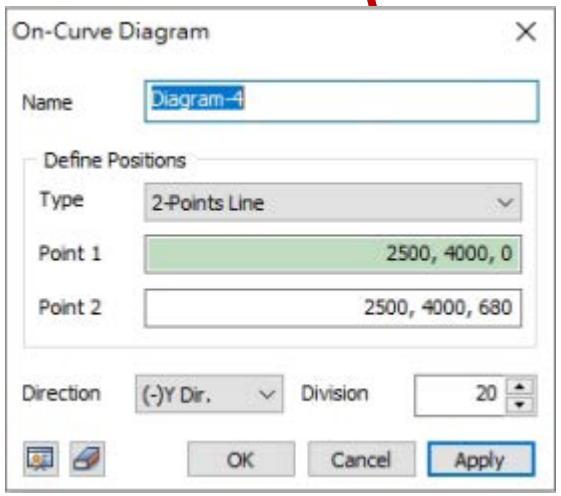


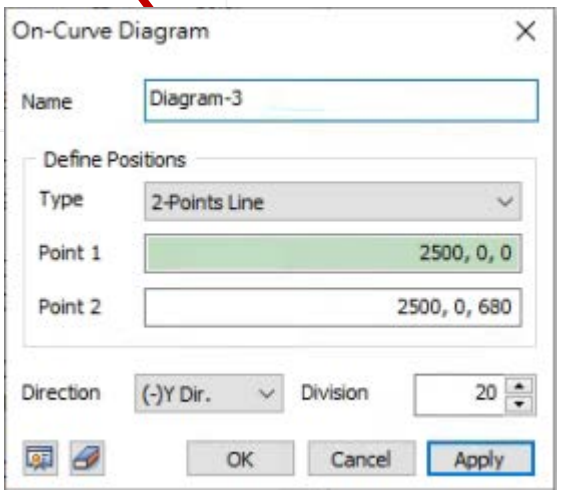
No.	X (m)	Y (m)	Z (m)	Value (...
1	2500	8396	0	0.000000e+000
2	2500	8396	34	1.000000e+001
3	2500	8396	68	1.099470e+001
4	2500	8396	102	1.167631e+001
5	2500	8396	136	1.531584e+001
6	2500	8396	170	1.560519e+001
7	2500	8396	204	1.565062e+001
8	2500	8396	238	1.582541e+001
9	2500	8396	272	1.650726e+001
10	2500	8396	306	1.728351e+001
11	2500	8396	340	1.743465e+001
12	2500	8396	374	1.747894e+001
13	2500	8396	408	1.758546e+001
14	2500	8396	442	1.797255e+001
15	2500	8396	476	1.865239e+001
16	2500	8396	510	1.863587e+001
17	2500	8396	544	1.871825e+001
18	2500	8396	578	1.890675e+001
19	2500	8396	612	1.918567e+001
20	2500	8396	646	1.941979e+001
21	2500	8396	680	1.951681e+001

輸出數值結果





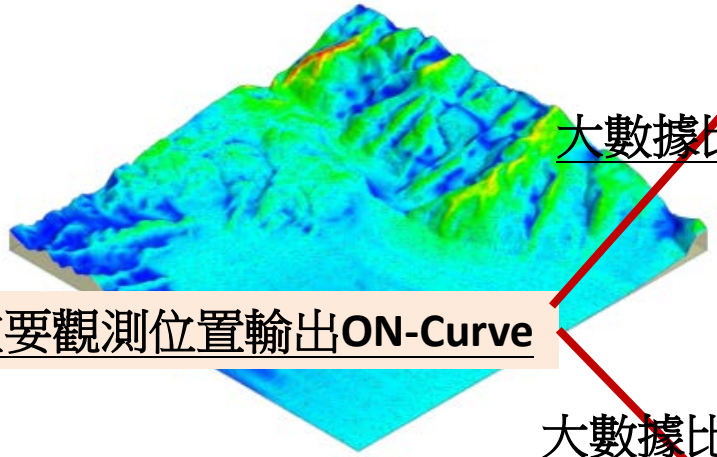




MIDAS

補充
(大數據整理)

地表風速分佈



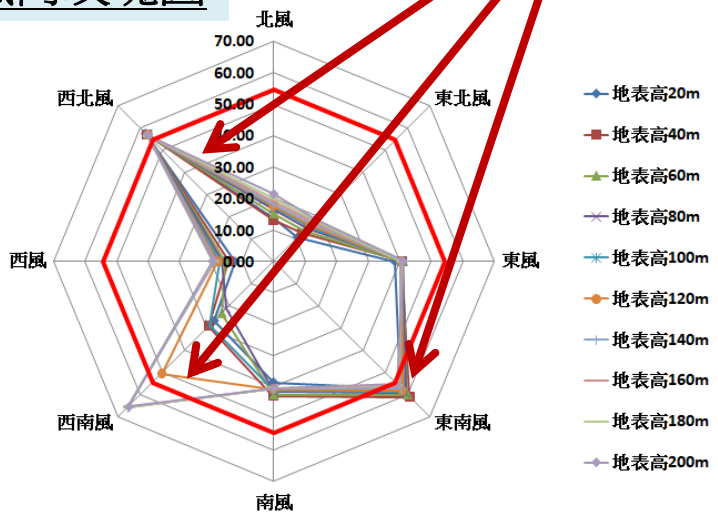
重要觀測位置輸出ON-Curve

大數據比對

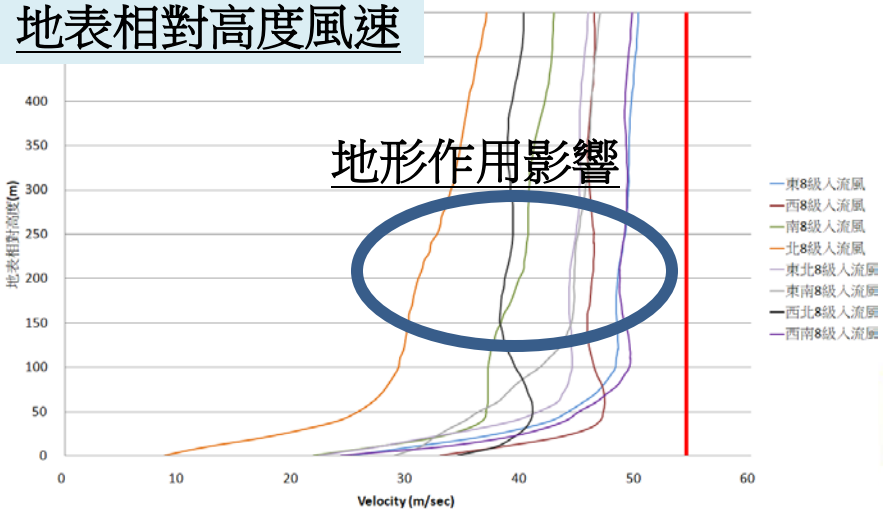
大數據比對

特別注意東南風/西南風/西北風

風向玫瑰圖



地表相對高度風速



midas NFX-CFD

CFD_標準教學系列

強制對流分析104-3D CAARC建築風場

台灣邁達斯

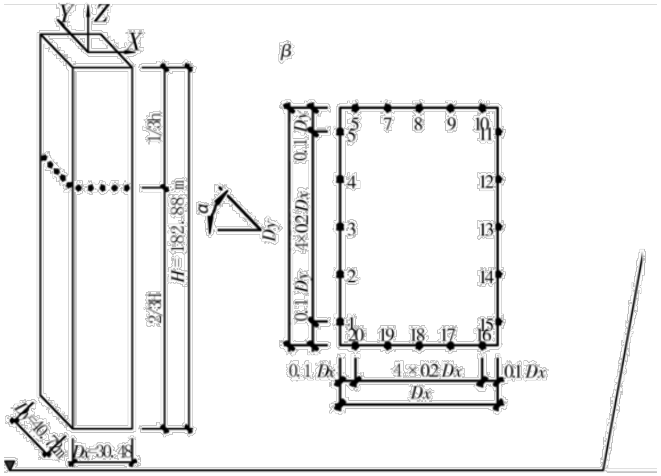
MIDAS

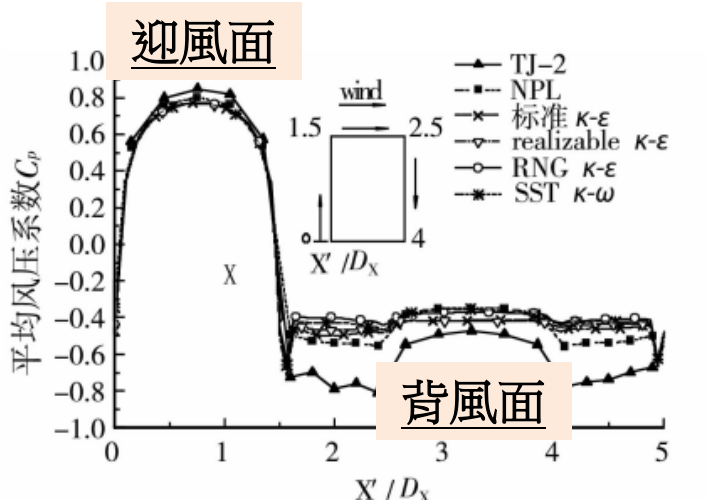
CAARC建築風場數值模擬-1

$$C_P = 2(P - P_0)/\rho U_0^2$$

C_p:風壓係數
P:觀測點平均壓力(通用標準測點取2/3H)
P_0:參考高度處的靜壓
ρ：空氣密度
U_0:建築物高度初始風速
H:建築物高度(m)

計算流域取600m×1800m×1000m)
(建築物40.72m×30.48m×182.88m置於流域沿流向前1/3處)





Reference
聶少鋒,周緒紅.CAARC標準高層建築三維鈍體繞流風場數值模擬.土木建築與環境工程,2009,Vol.31 No.6

CAARC建築風場數值模擬-2

$$U = U_0 \left(\frac{Z}{Z_0}\right)^{\alpha}$$

U:相對高度風速(m/s)
Z:地表相對高度(m)
Z_0:參考高度(常見取地表10m高)
U_0:參考高度風速(m/s)
α :地面粗糙度指數

类别	描述	Z_G/m	α
A	指近海海面、海岛、海岸、湖岸及沙漠地区	300	0.12
B	指田野、乡村、丛林、丘陵及房屋比较稀疏的乡镇和城市郊区	350	0.16
C	指有密集建筑群的城市市区	400	0.22
D	指有密集建筑群且房屋 较高的城市市区	450	0.30

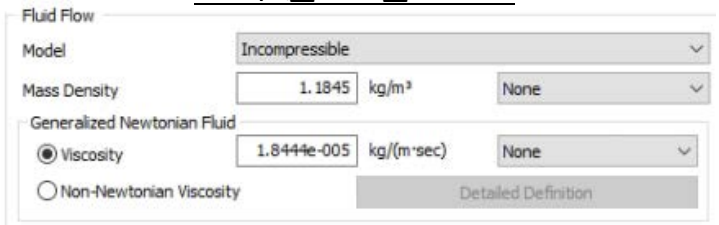
Reference
聶少鋒,周緒紅.CAARC標準高層建築三維鈍體繞流風場數值模擬.土木建築與環境工程,2009,Vol.31 No.6

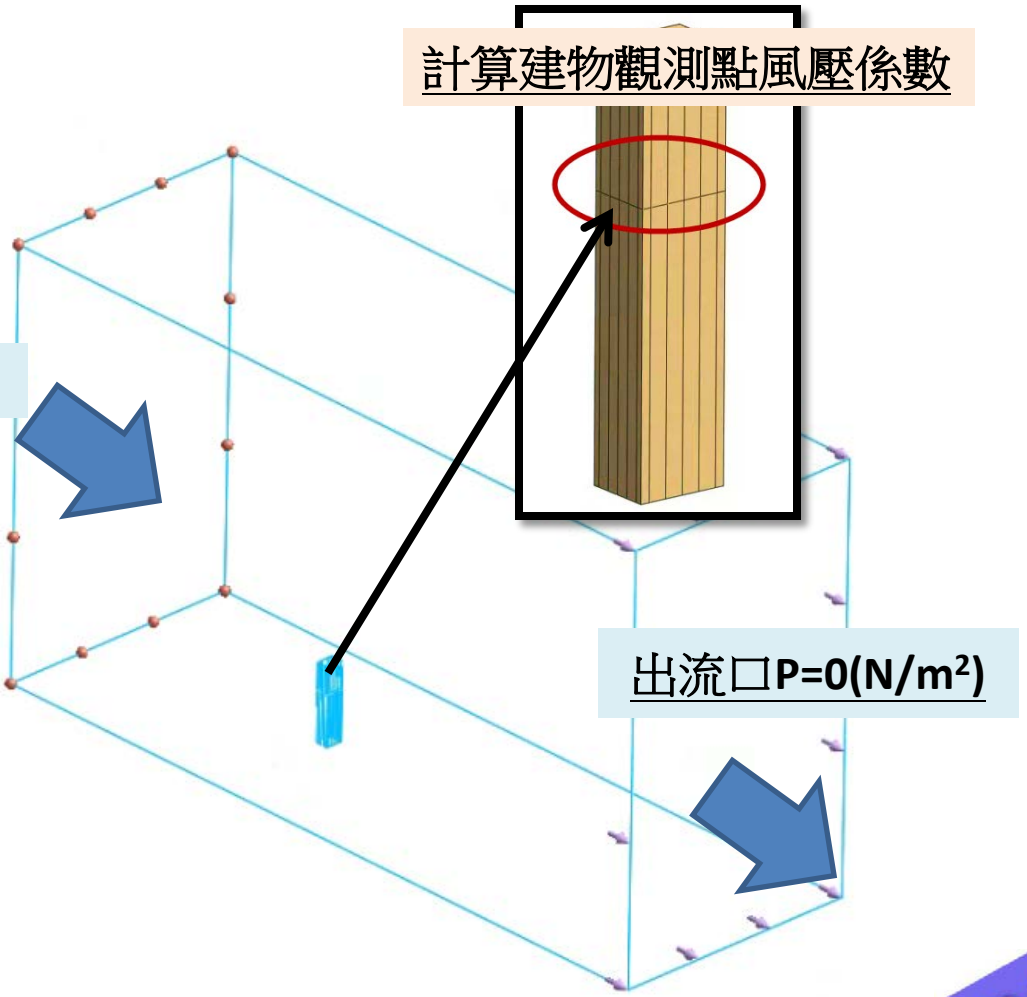
分析說明

Wall Function(外側壁面)

(Dimensionless Wall Distance 地表y+=65/建物y+=35)

空氣_AIR_25 ℃





$$u = u_r \left(\frac{z}{z_r} \right)^{\alpha}$$

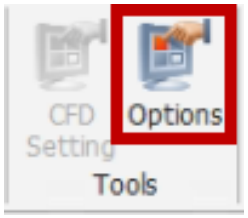
u:相對高度風速(m/s)

Z_γ:地表高度(10m)

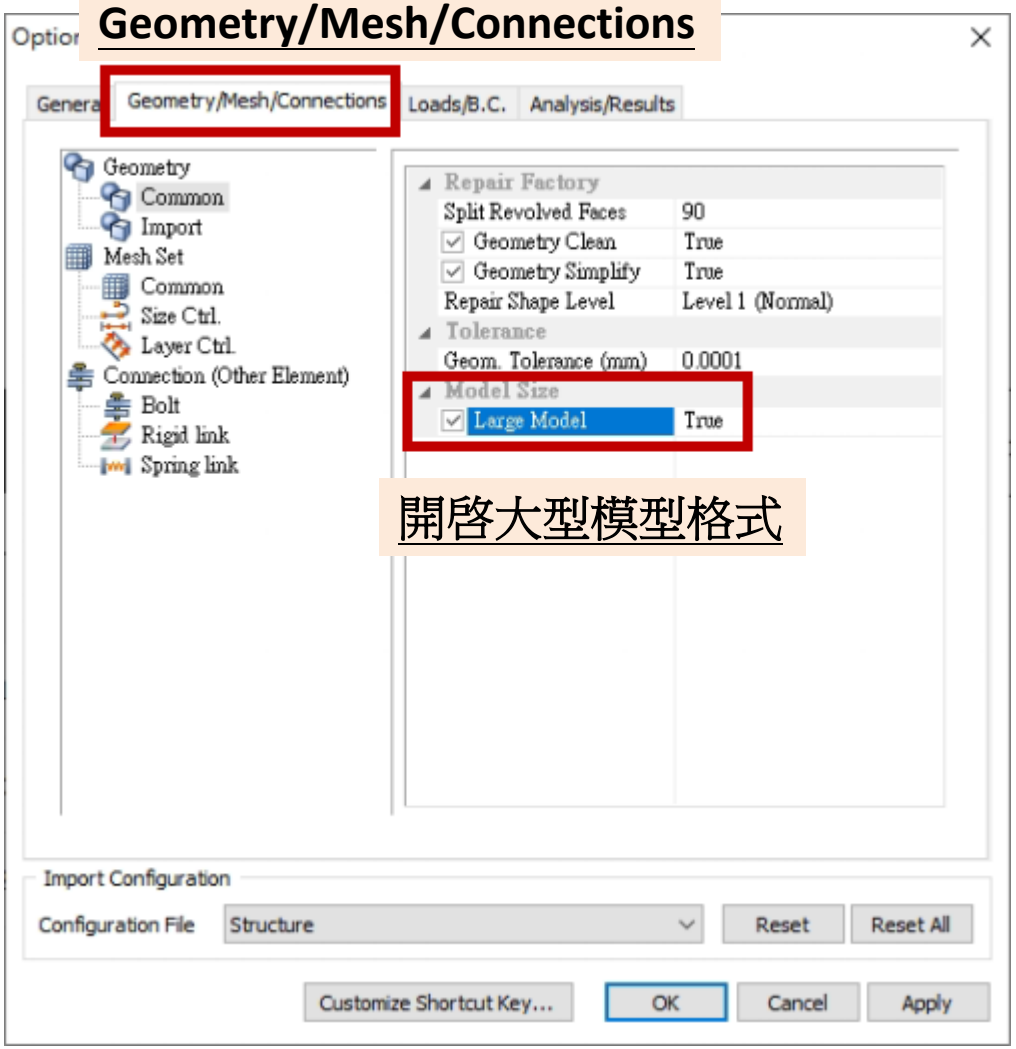
u_γ:地表高度Z_γ風速(假設10m/s)

α :0.3

大型模型



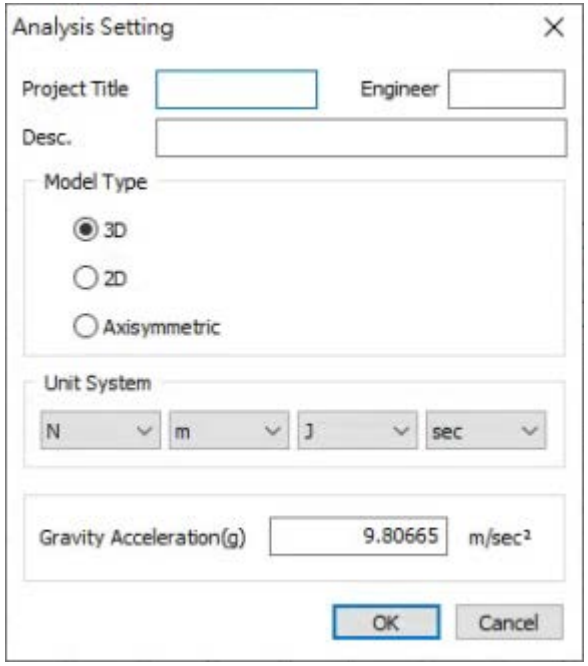
Geometry/Mesh/Connections



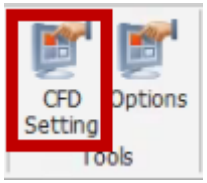
開啓大型模型格式

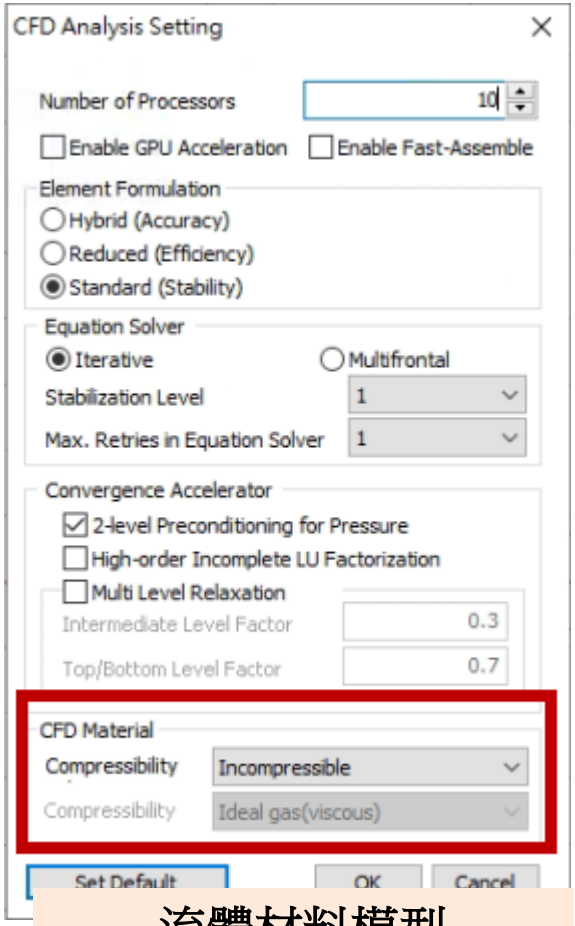
註:大型模型格式,匯出幾何格式會自動調整尺寸比例1/1000。

環境



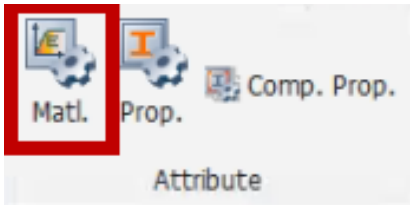
單位使用N/m/J/sec

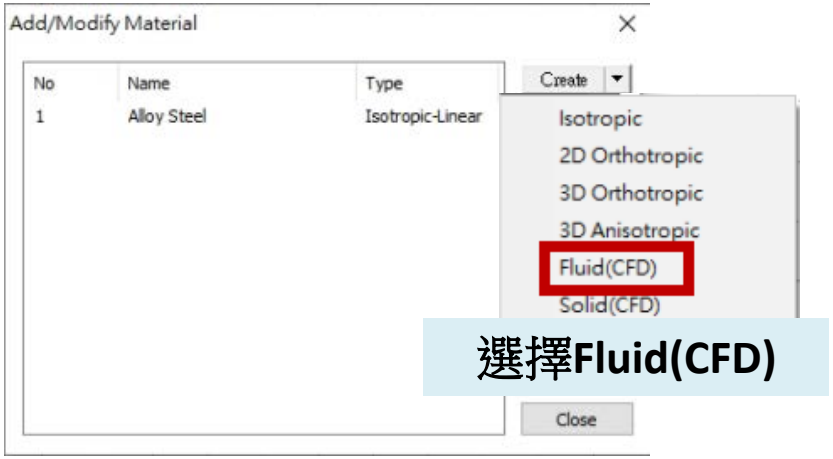




流體材料模型
不可壓縮流

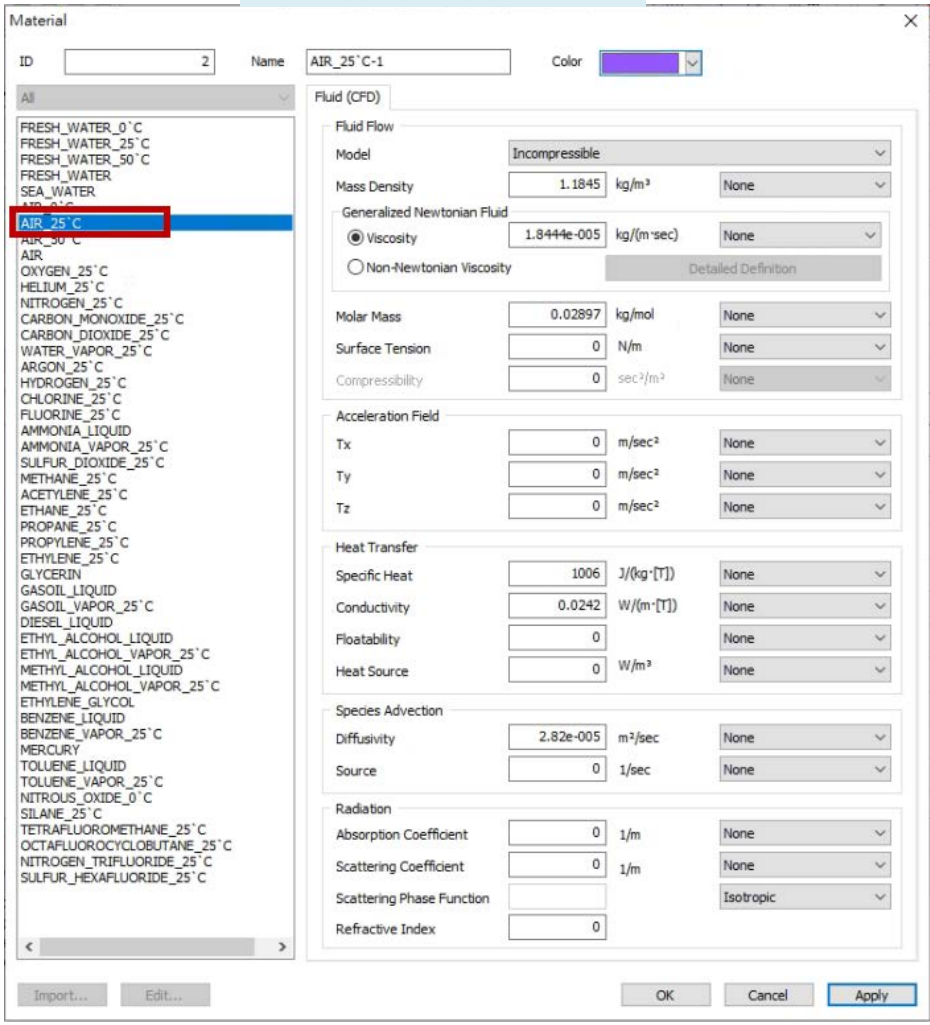
材料



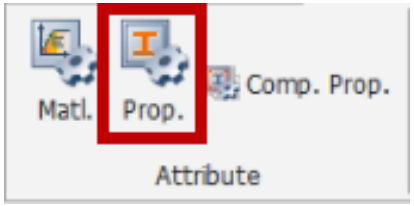


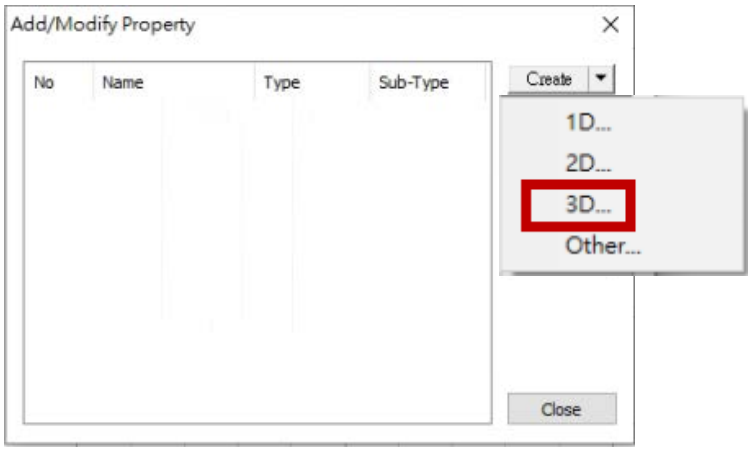
選擇Fluid(CFD)

新增AIR 25 ℃

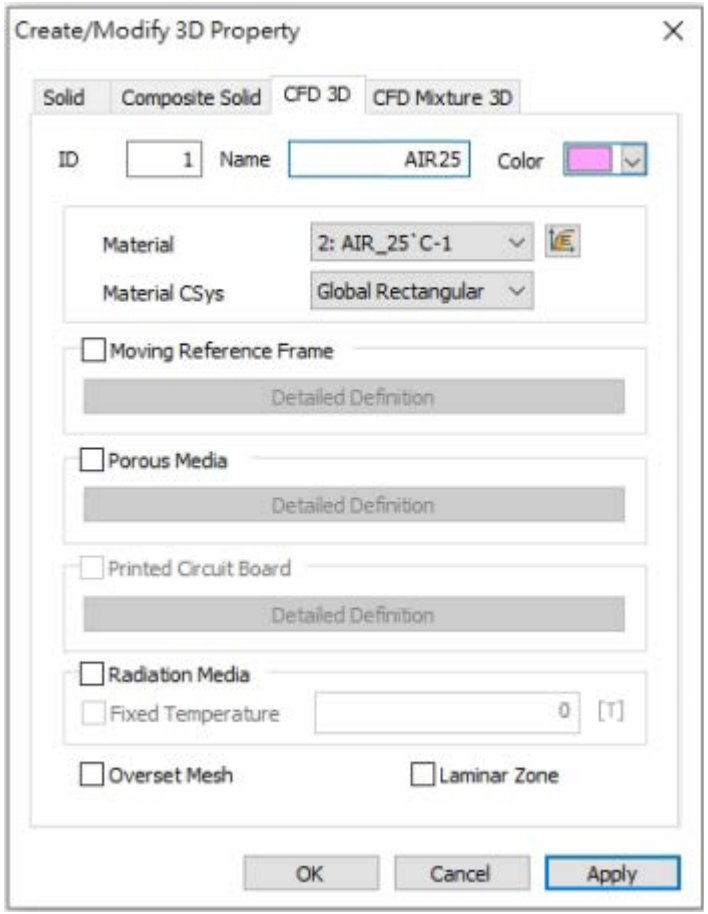


屬性

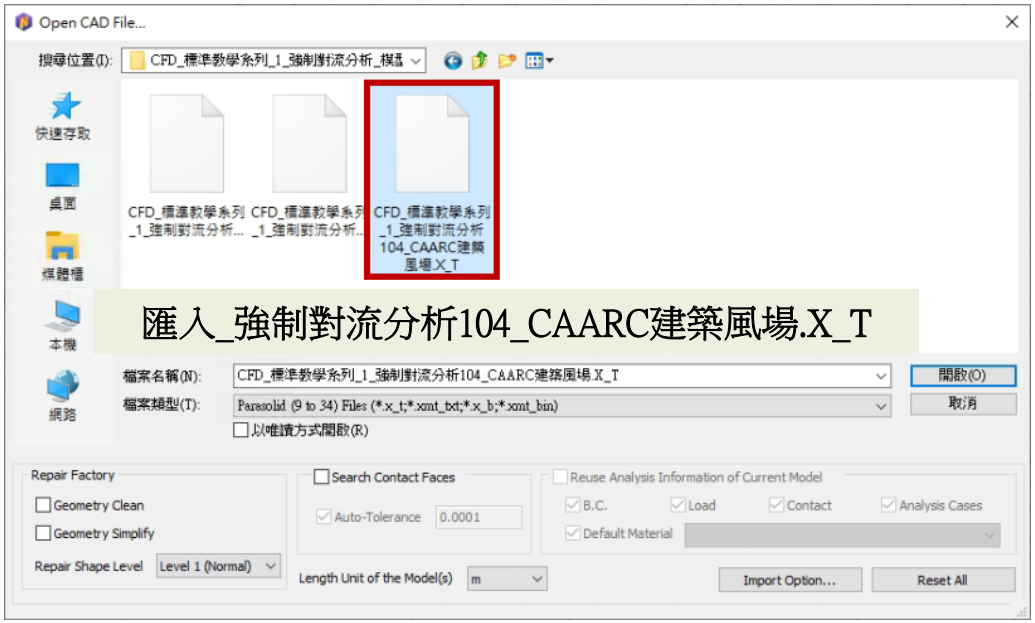




新增AIR 25 ℃ Property

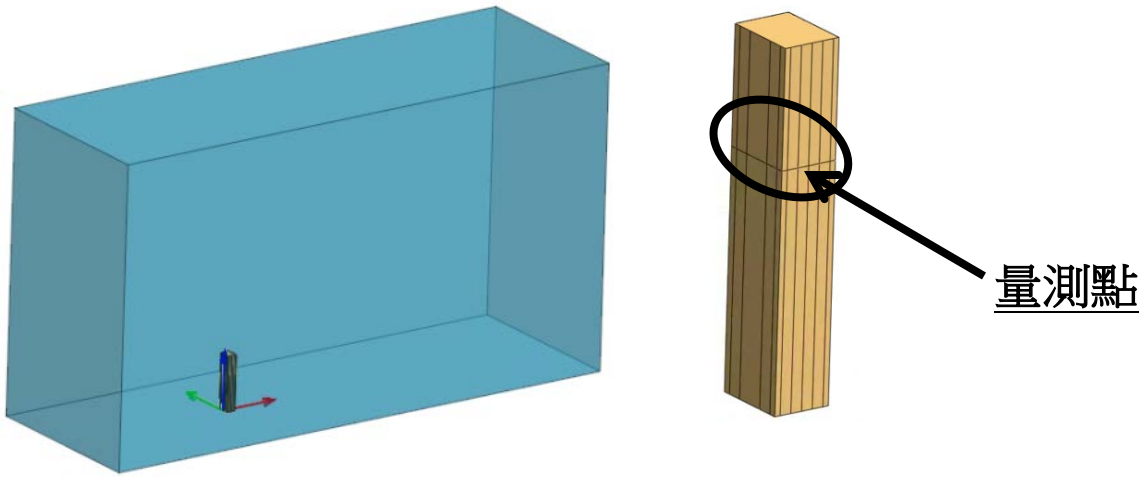


匯入模型

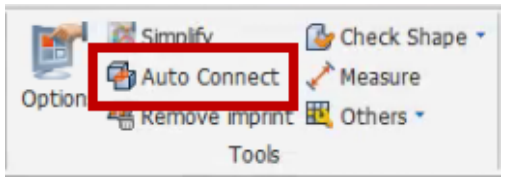


計算流域取600m×1800m×1000m)

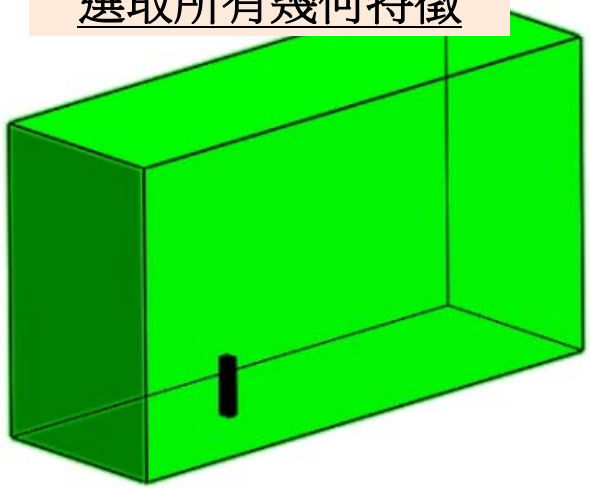
(建築物40.72m×30.48m×182.88m置於流域沿流向前1/3處)

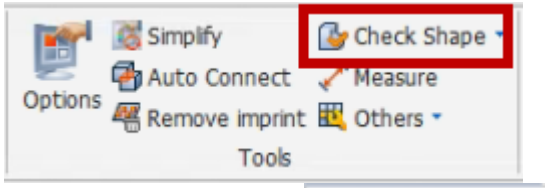


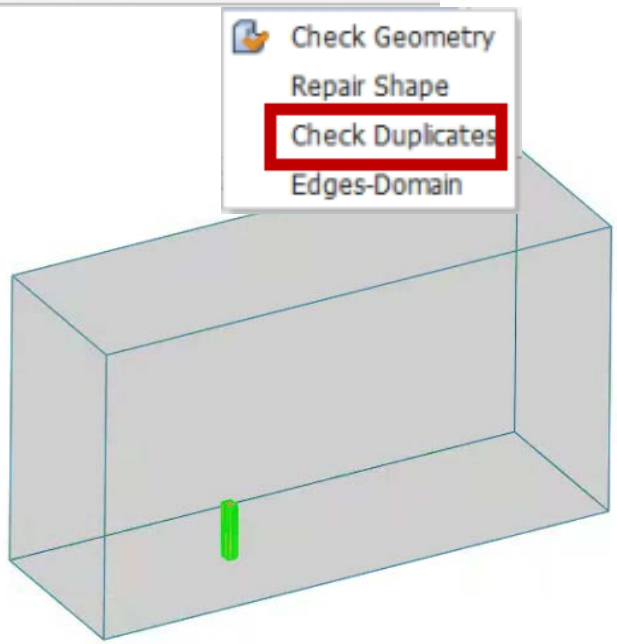
相鄰面特徵建立&確認



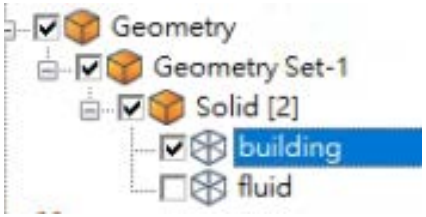
選取所有幾何特徵



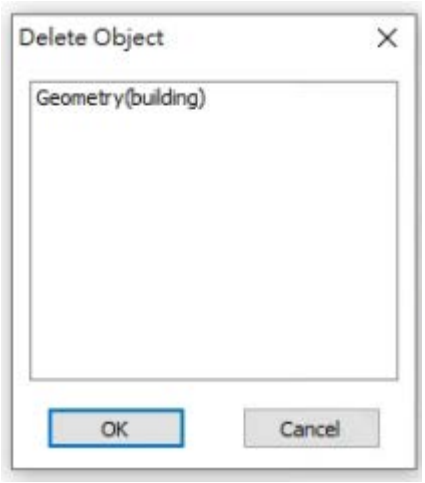




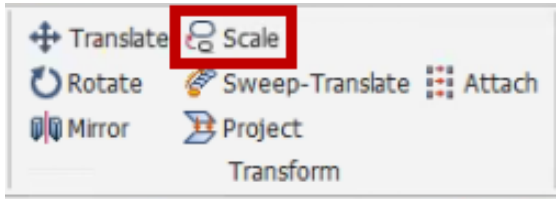
刪除特徵

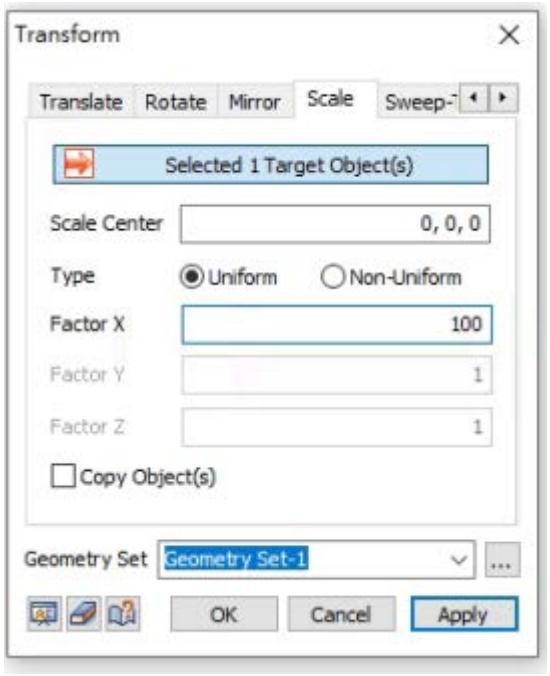


鍵盤Delete鍵

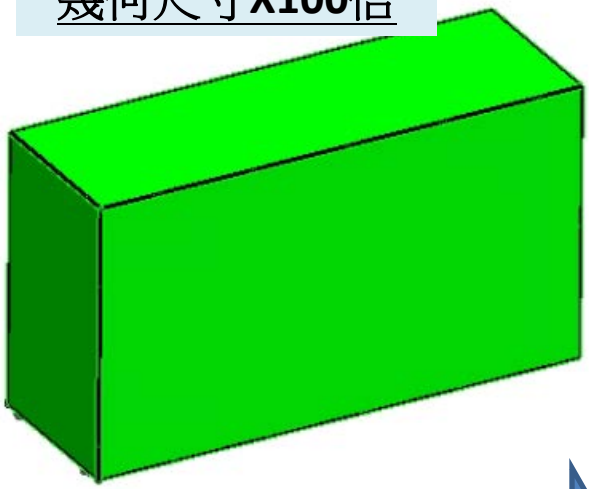


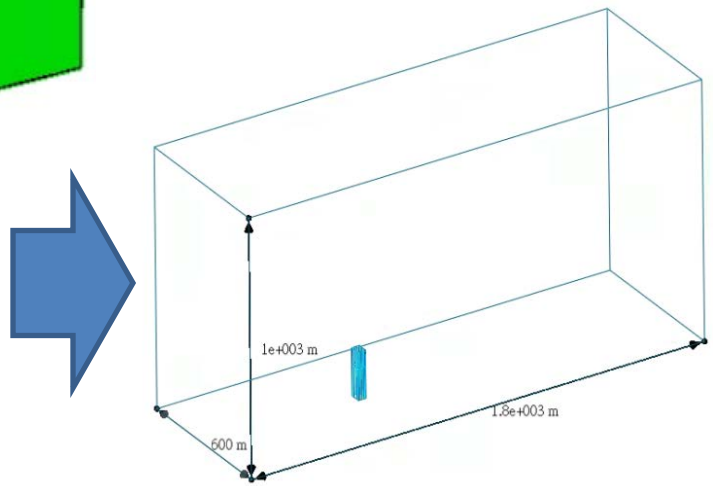
幾何尺寸放大



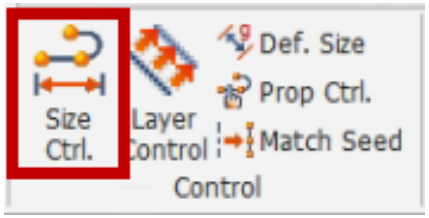


幾何尺寸**X100倍**

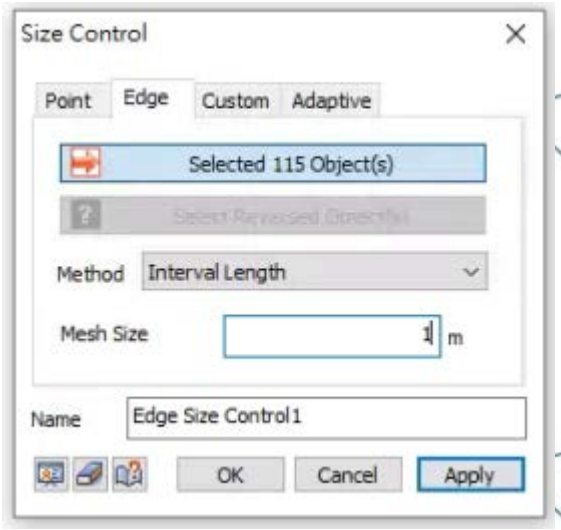


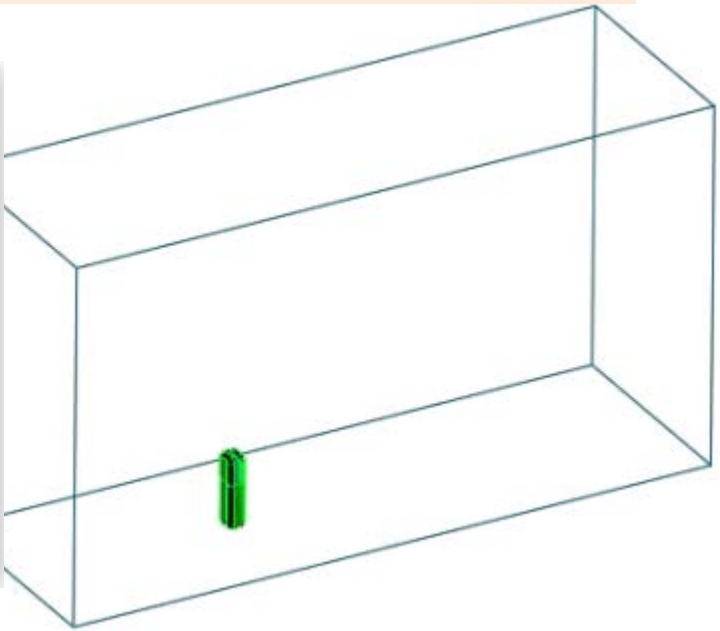


手動網格尺寸



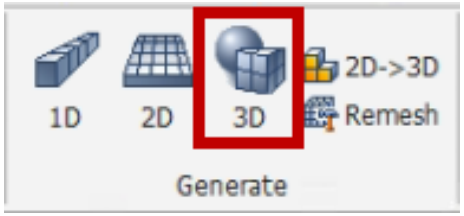
流場幾何建物特徵指定網格尺寸**1(m)**



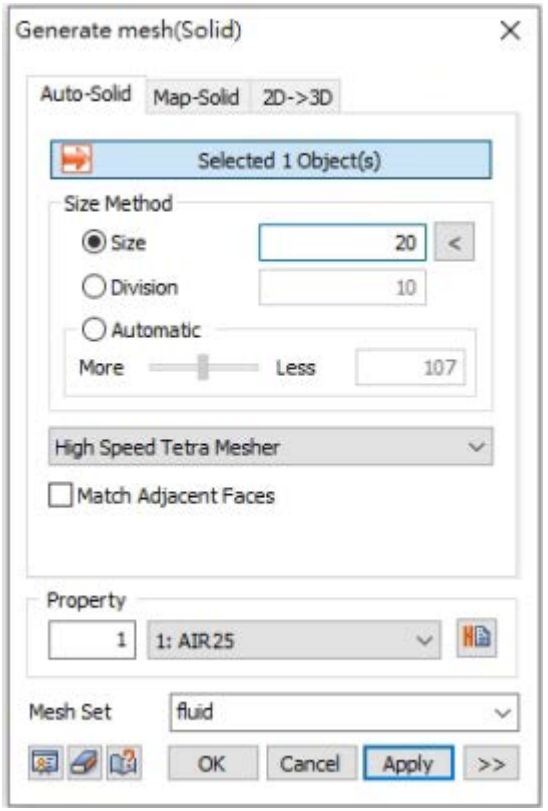


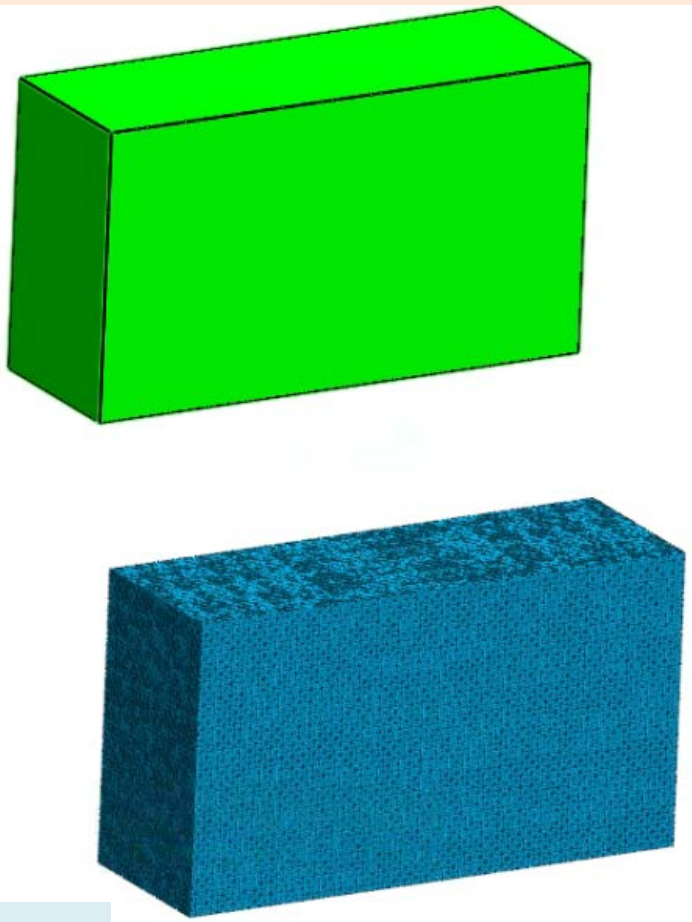
midas NFX-CFD

網格



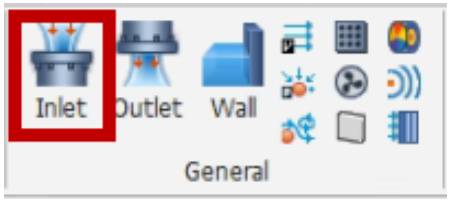
Tetra Mesher(四面體網格)/網格尺寸20 (m)/1階元素





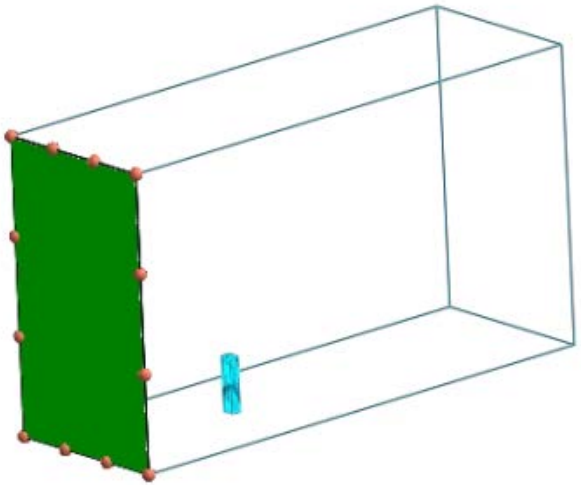
NFX CFD不支援二階(高階)元素

入流

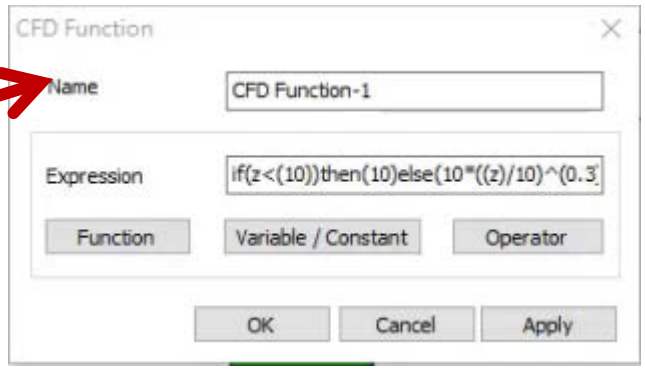


入流流速-函數定義

if(z<(10))then(10)else(10*((z)/10)^(0.3))endif



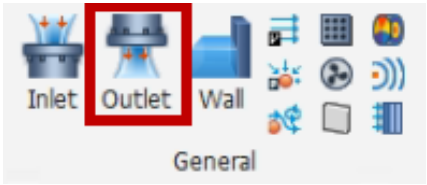
Step1.新增風廓線入流函數

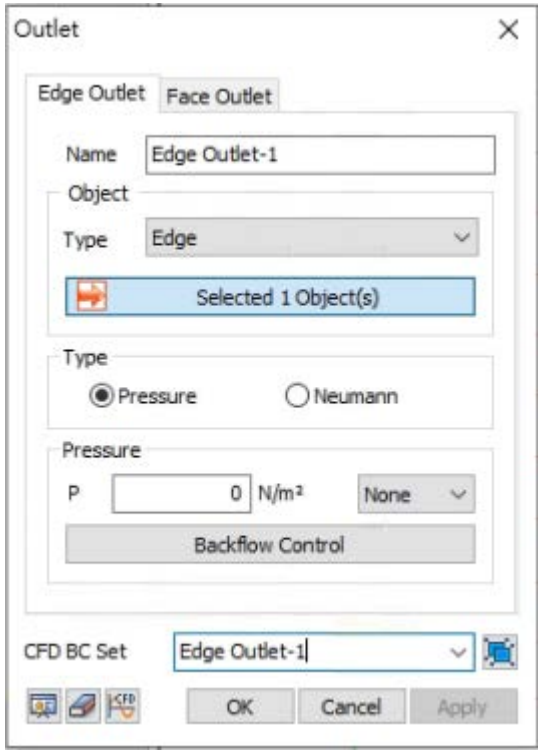


Step2.指定函數

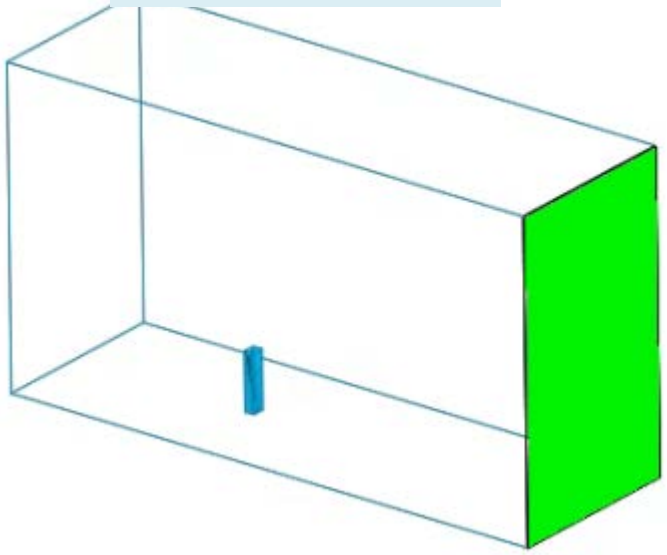
註:不同邊界&載荷建議使用不同集合名稱。

出流

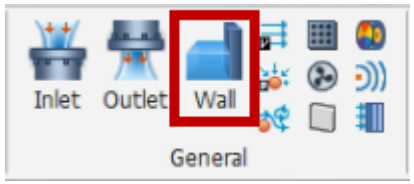




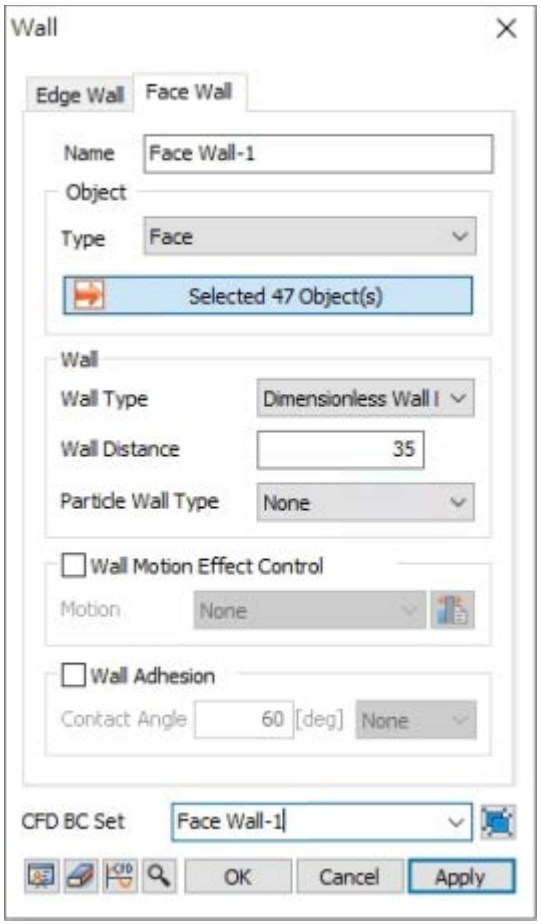
壓力0(N/m²)

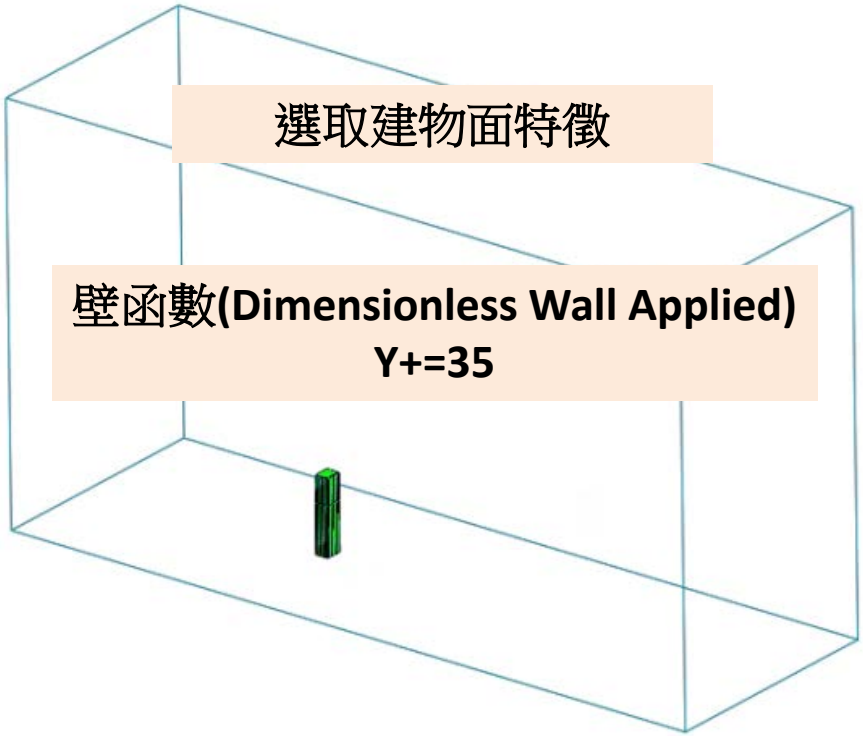


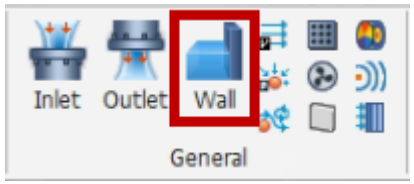
註:不同邊界&載荷建議使用不同集合名稱。



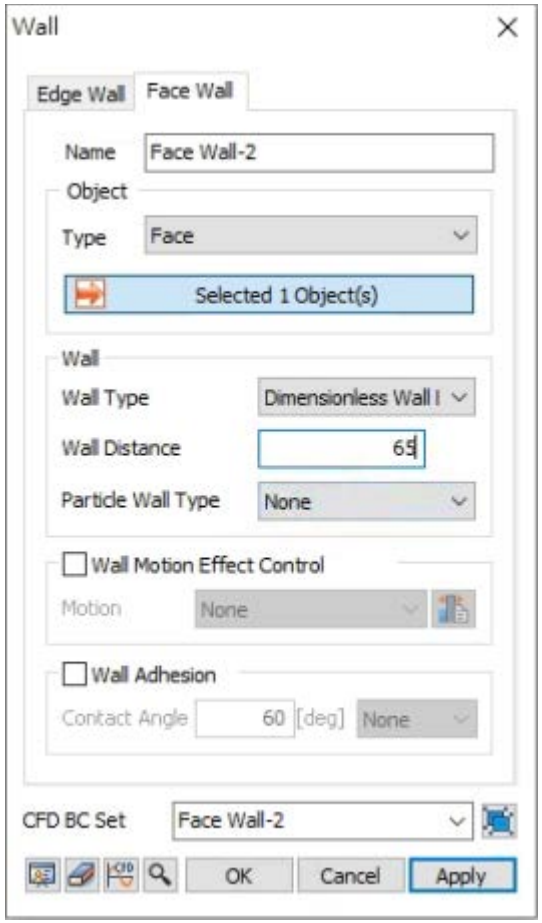
壁函數-1
(Dimensionless Wall Applied)





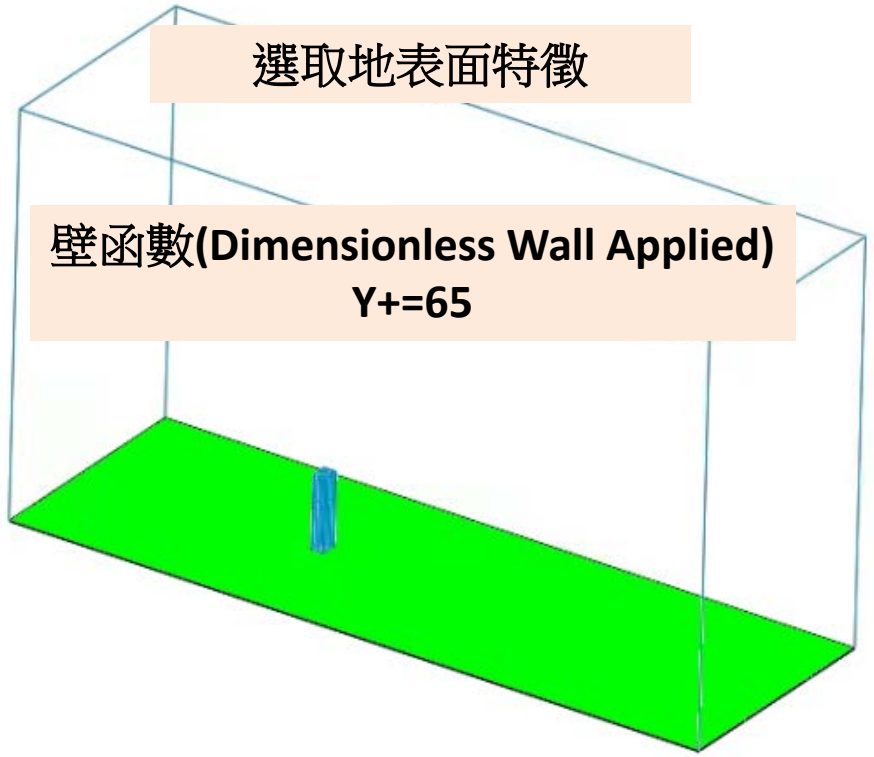


壁函數-2
(Dimensionless Wall Applied)

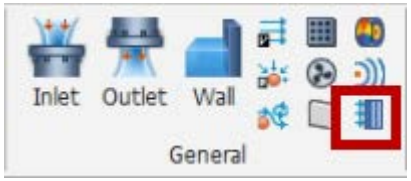


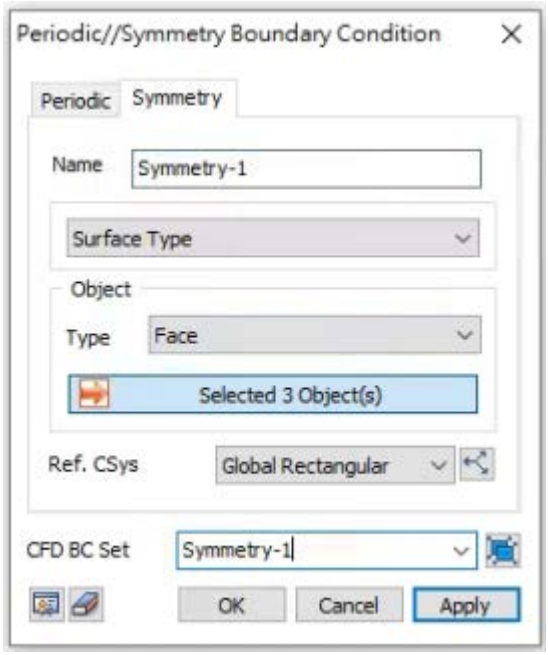
選取地表面特徵

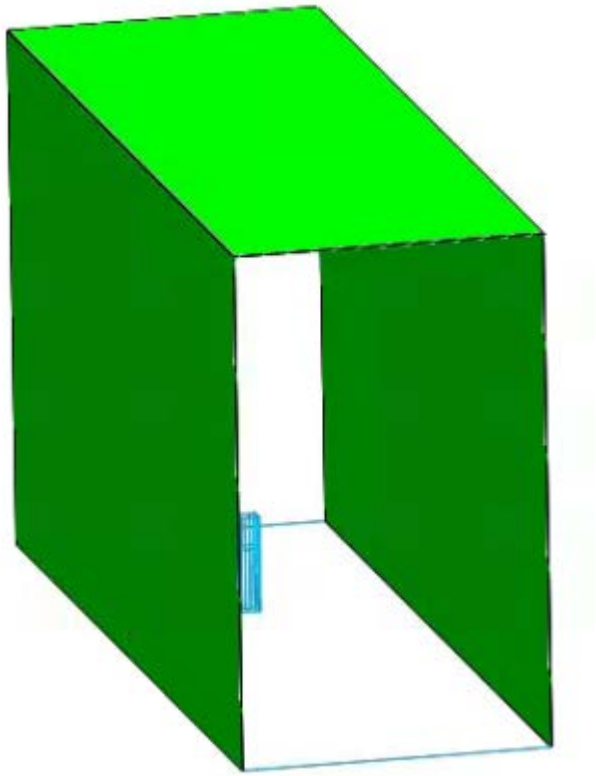
壁函數(Dimensionless Wall Applied)
Y+=65



對稱

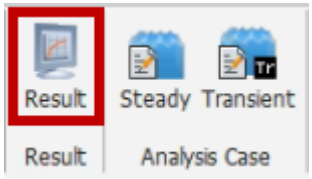




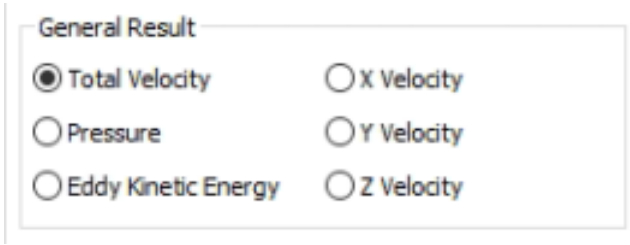


註:不同邊界&載荷建議使用不同集合名稱。

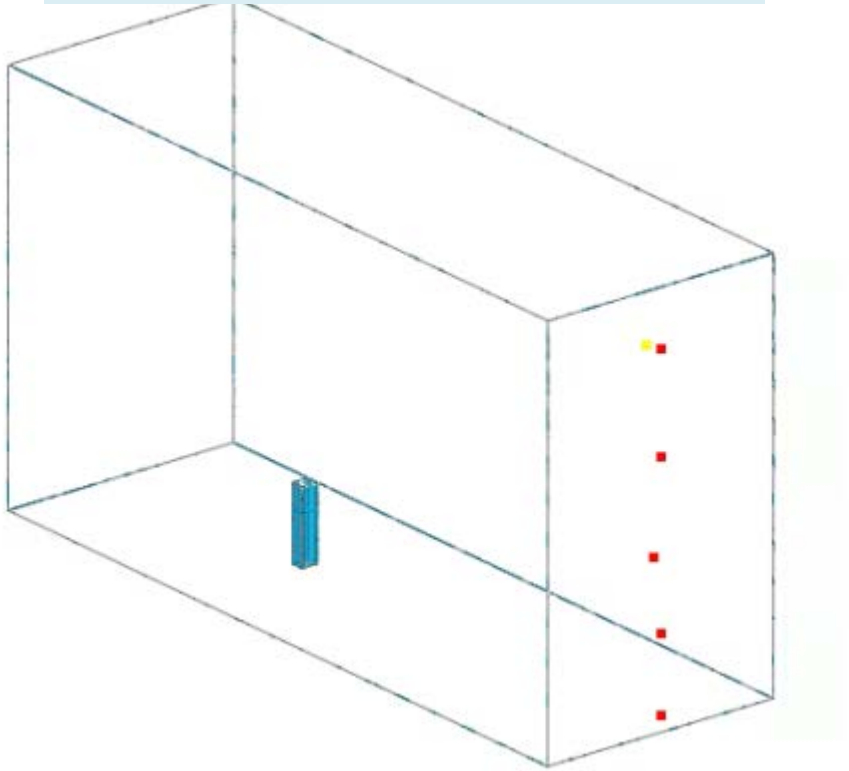
觀測點

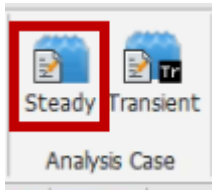


輸出特定節點(速度)數據



選取出流口節點作為觀測節點

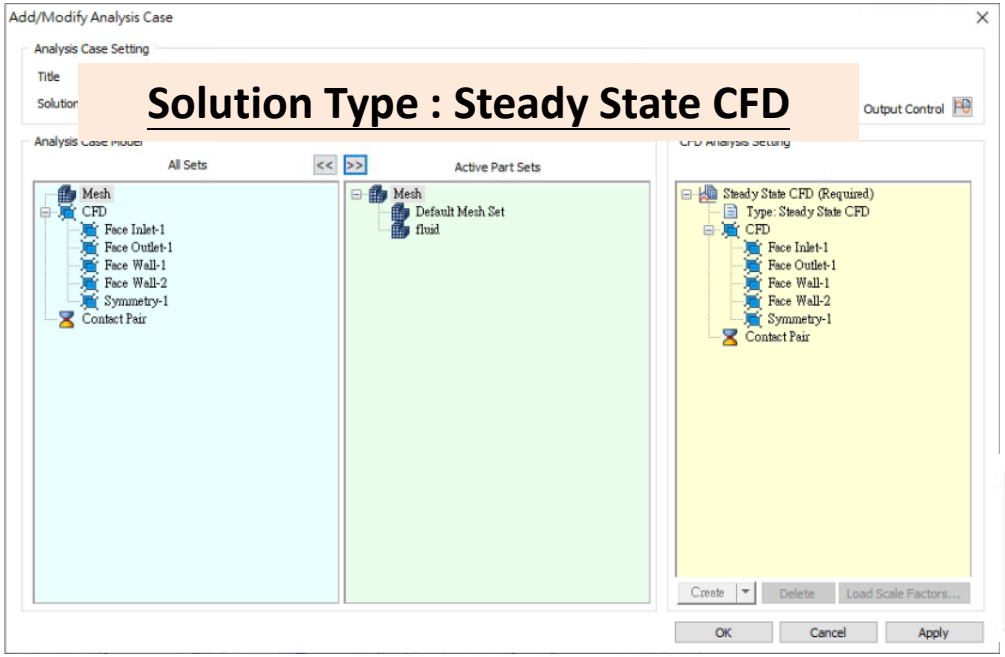


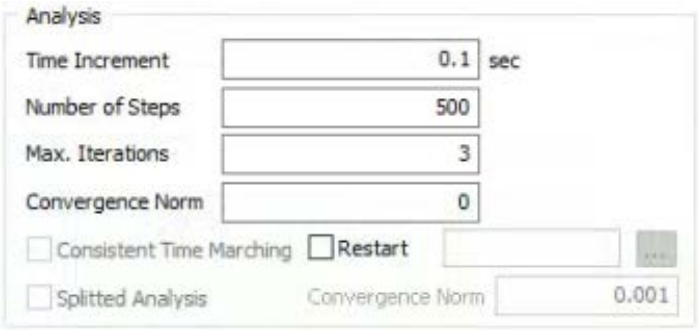


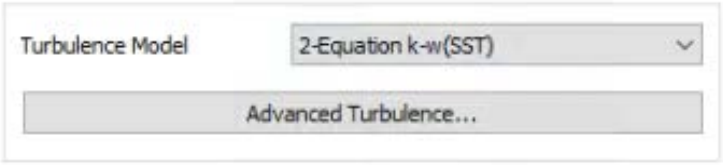
CFD穩態分析

(CAARC建築風場數值模擬)

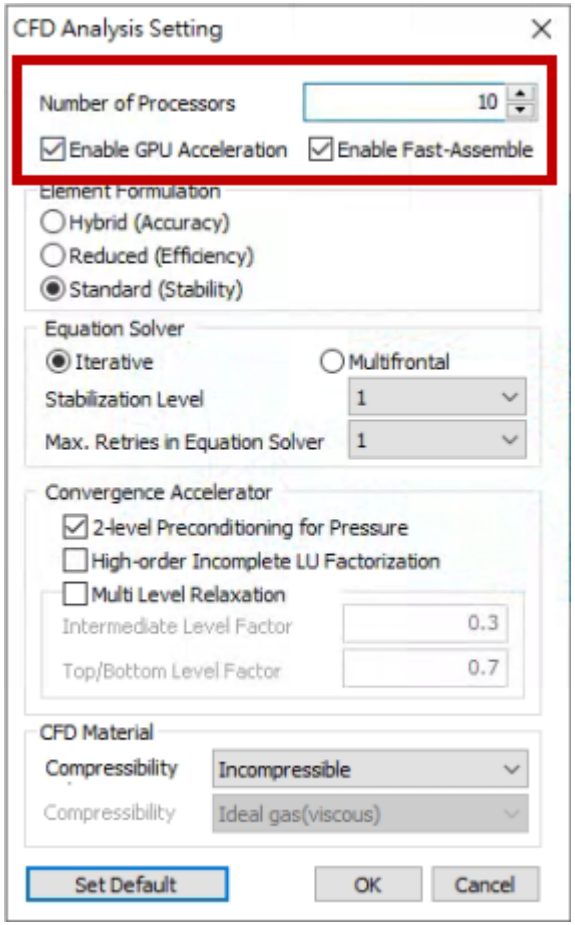
增量步數目不夠，無法得到合理風壓數據







CPU&GPU定義

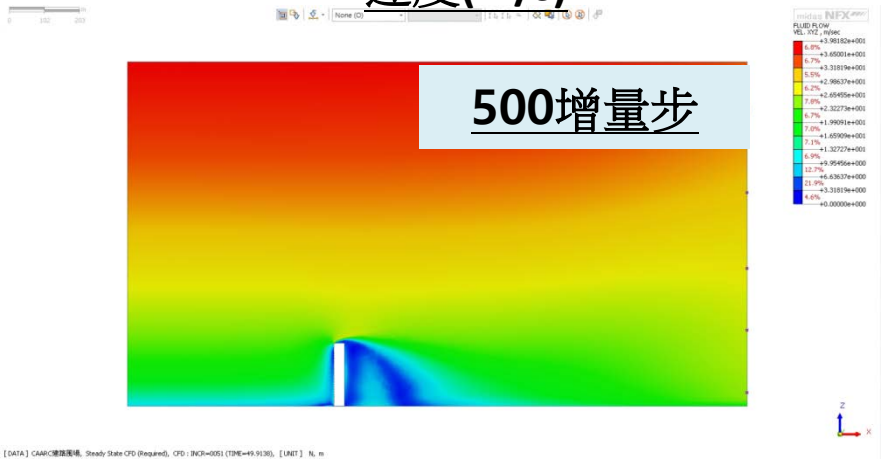

CFD Analysis Setting
Number of Processors
10
Enable GPU Acceleration
Enable Fast-Assemble
Element Formulation
Hybrid (Accuracy)
Reduced (Efficiency)
Standard (Stability)
Equation Solver
Iterative
Multifrontal
Stabilization Level
1
Max. Retries in Equation Solver
1
Convergence Accelerator
2-level Preconditioning for Pressure
High-order Incomplete LU Factorization
Multi Level Relaxation
Intermediate Level Factor
0.3
Top/Bottom Level Factor
0.7
CFD Material
Compressibility
Incompressible
Compressibility
Ideal gas(viscous)
Set Default
OK
Cancel

分析結果-1

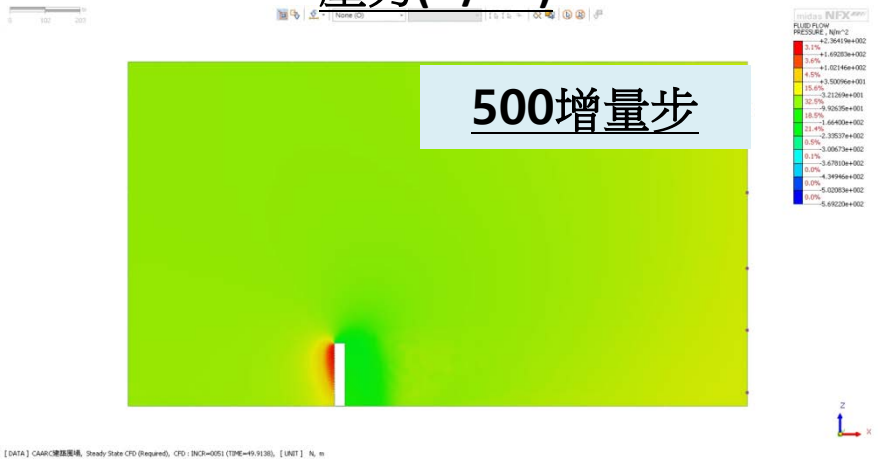
(輪廓剖面圖)

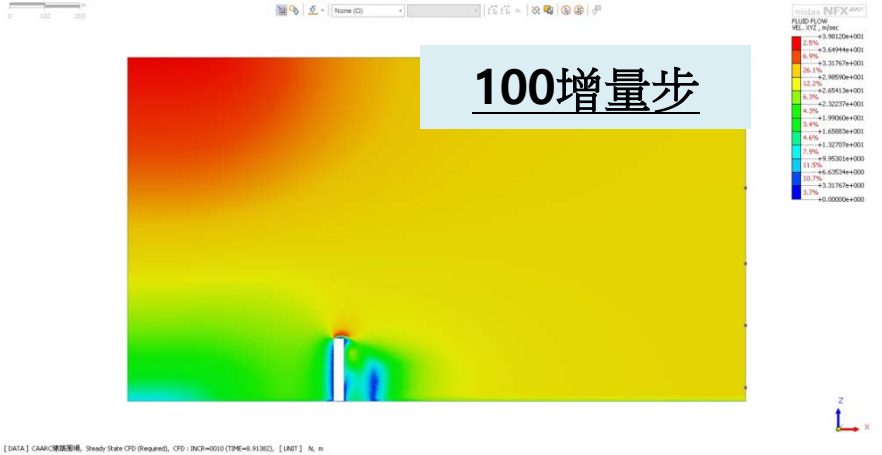
增量步數目不夠，無法得到合理風壓數據

速度(m/s)



壓力(N/m²)

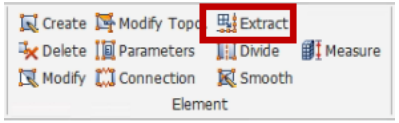


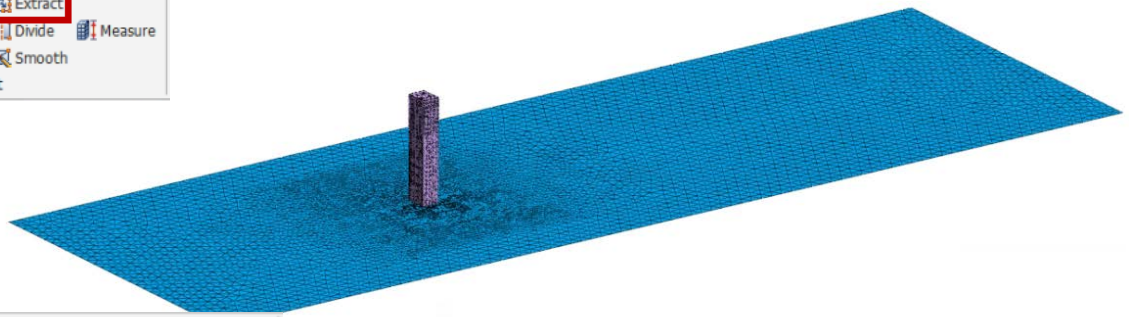


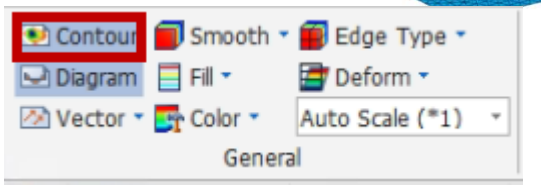
分析結果-2

(500增量步)

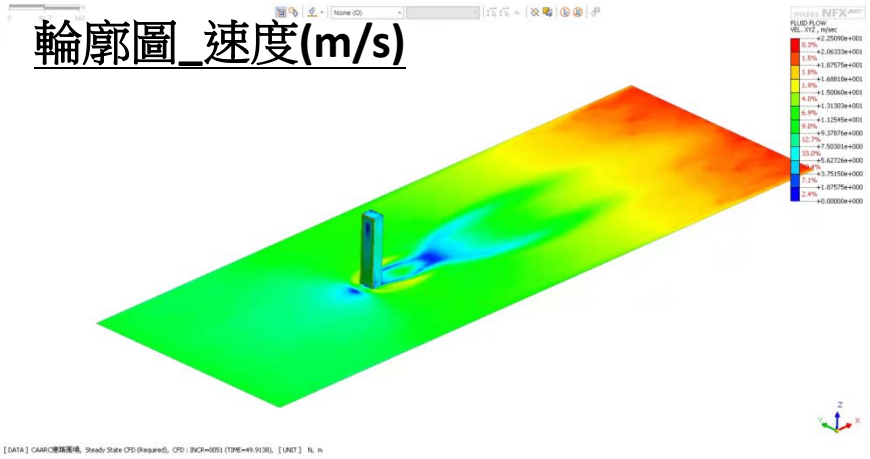
提取建物和地表2D元素



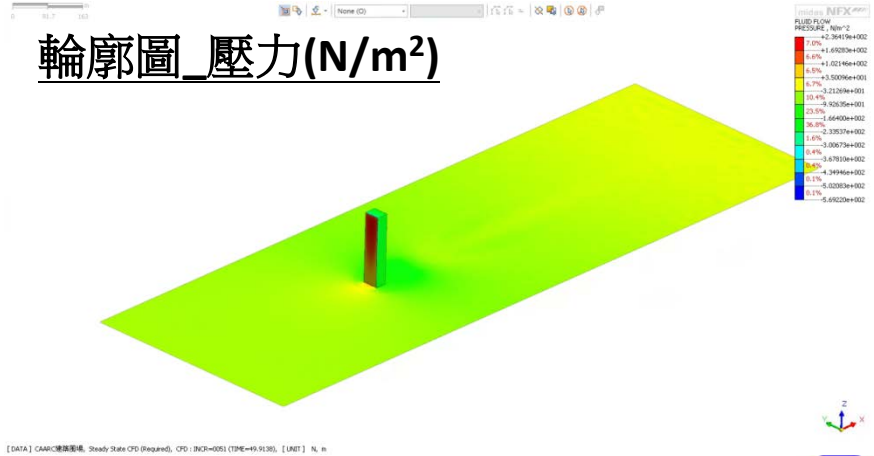




輪廓圖_速度(m/s)



輪廓圖_壓力(N/m²)

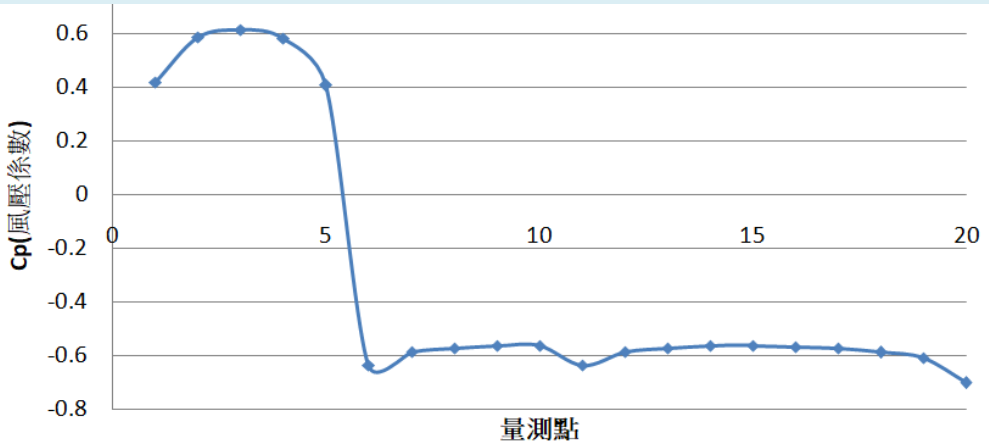


分析結果-3

(500增量步)

計算參考Excel

範例取第500增量計算, 建議取多增量步平均值



數值結果查詢