

仍然會顯示有反力呢?

可能有兩個原因導致在無支撐的節點上有反力。

1. Structure Type

在Structure type的選項中選擇了 X-Z Plane、 Y-Z Plane 或 X-Y plane,但載重是施加在第三方向上。舉例來說,若選擇了 X-Y plane 且載重是沿著Z方向施加,則所有的節點在Z方向都會有反力,因為在Z方向被束制了。因此,若要施加多維的載重,務必要選擇 3-D作為Structure Type。此設定可以從 Structure > Structure Type 進行設定。

為什麼沒有在節點上設定支撐(support),

			X			
Structure Type						
3-D	Y-Z Plane	X-Y Plane	Constraint RZ			
Mass Control Parameter						
Lumped Mass						
Consider Off-diagonal Masses						
Considering Rotational Rigid Body Mode for Modal Participation Factor Consistent Mass						
Convert Self-weight int	o Masses					
Convert Self-weight int	D Masses Z O Convert	to X, Y 🤇) Convert to Z			
Convert Self-weight int	o Masses Z 💿 Convert	to X, Y 🖉) Convert to Z			
Convert Self-weight int Convert to X, Y, Gravity Acceleration :	o Masses Z O Convert 9.806 m/s	to X, Y 🤇) Convert to Z			
Convert Self-weight int	9.806 m/s	to X, Y () ec^2) Convert to Z			
Convert Self-weight int Convert to X, Y, Gravity Acceleration : Initial Temperature :	0 Masses Z O Convert 9.806 m/s 0 [C]	to X, Y 🤇 ec^2) Convert to Z			
Convert Self-weight int Convert to X, Y, Convert to X, Y, Conve	0 Masses Z O Convert 9,806 m/s 0 [C] 0 with Center Line (X-Y	to X, Y @ ec^2 Plane) for Display) Convert to Z			
Convert Self-weight int Convert to X, Y, Convert to X, Y, Conve	2 Convert 9.806 m/s 0 [C] a with Center Line (X-Y ection with Center Line	to X, Y) Convert to Z			
Convert Self-weight int Convert to X, Y, Convert to X, Y, Cravity Acceleration : Initial Temperature : Align Top of Beam Section Align Top of Slab(Plate) S	o Masses Z Convert 9.806 m/s 0 [C] o with Center Line (X-Y ection with Center Line	to X, Y ec^2 Plane) for Display e (X-Y Plane) for Disp) Convert to Z			
Convert Self-weight int Convert to X, Y, Convert to X, Y,	o Masses Z Convert 9.806 m/s 0 [C] o with Center Line (X-Y ection with Center Line	to X, Y ec^2 Plane) for Display e (X-Y Plane) for Disp	Convert to Z			

MIDAS

2. 沉陷的載重無設置邊界條件修正

沉陷的載重有被施加在模型中,但其邊界條件的轉換設定卻並未隨之調整。因此, 所有的靜載重在完工階段,沉陷都會先被計算,然後才會執行靜力分析。 Boundary Change Assignment可以參考下圖完成設定。路徑為: Analysis → Boundary Change Assignment

Boundary Change Assignment to Load Cases/Analyses						
Boundary Change Assignment to Load Case Pata Selection Support Point Spring Support Change General Link Property Section Stiffness Scale Factor Plate Stiffness Scale Factor Beam End Release	s/Analyses Boundary Group Combination Boundary Group List Temp span 1 Temp span 2 Permanent Filter Boundary Group Filter Boundary Group	3 Name Settlement B-G Comb Add/Replace Delete Name Boundary Group Settlement B-G Comb permanent				
Load Cases & Analyses		Boundary Group Combination				
4 oving load Analysis		Settlement P.C.Comb	_			
Time History/Response Spectrum/Eigenvalue Analysis		Unchanged				
Pushover Analysis		Unchanged	=			
Time History Nonlinear Static Analysis		Unchanged				
Unlisted Analysis Types		Unchanged				
5			•			
I Constrain DOF associated with Specified Displacements/Settlements by Boundary Group Combinations						
Data Selection : Only Select the Data that Need to be Changed for Different Load Case 6 Remove Boundary Change Assignment to Load Cases/Analyses 0K						

- 1. 選擇所有和support相關的資訊
- 2. 選擇所有永久性邊界條件群組
- 3. 將所有邊界條件群組組合取名,並點選Add/Replace
- 4. 選擇所有沉陷分析會使用到的邊界條件群組組合
- 5. 將 Constraining DOF associated with settlement 選項打勾
- 6. 點選 OK 已完成設置