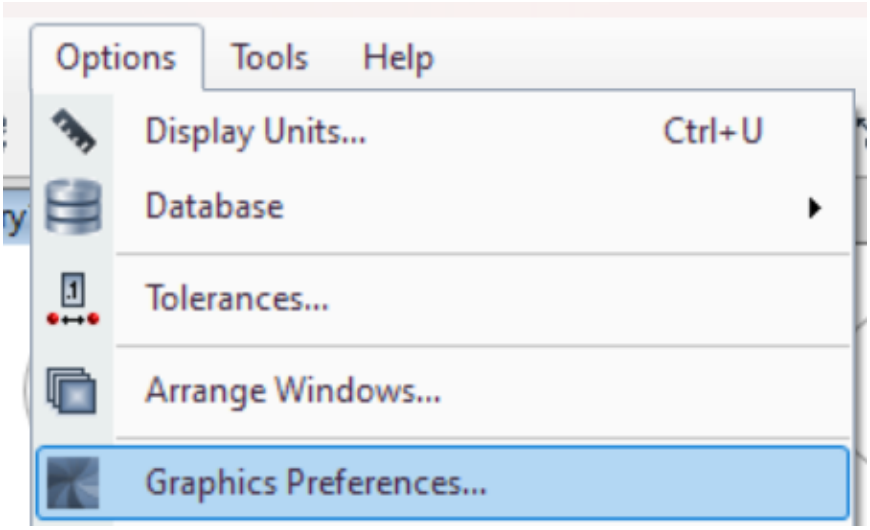
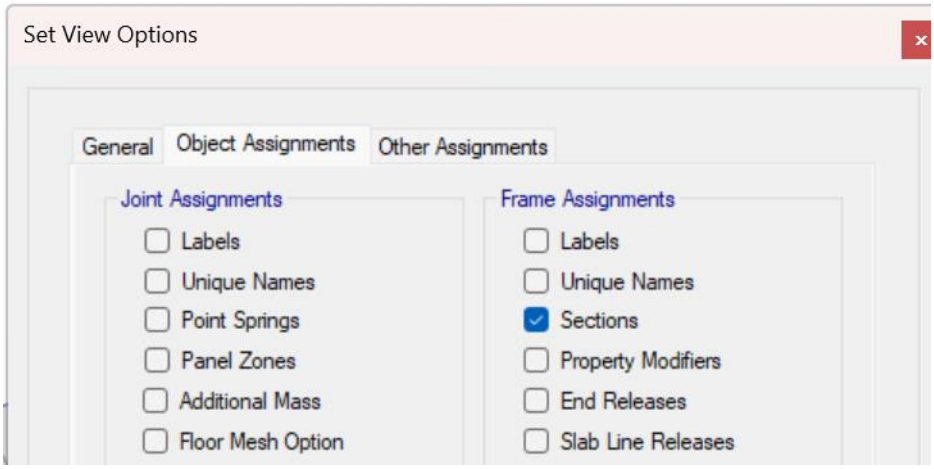
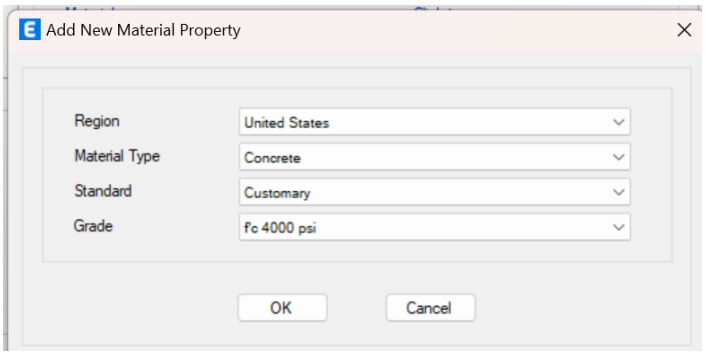


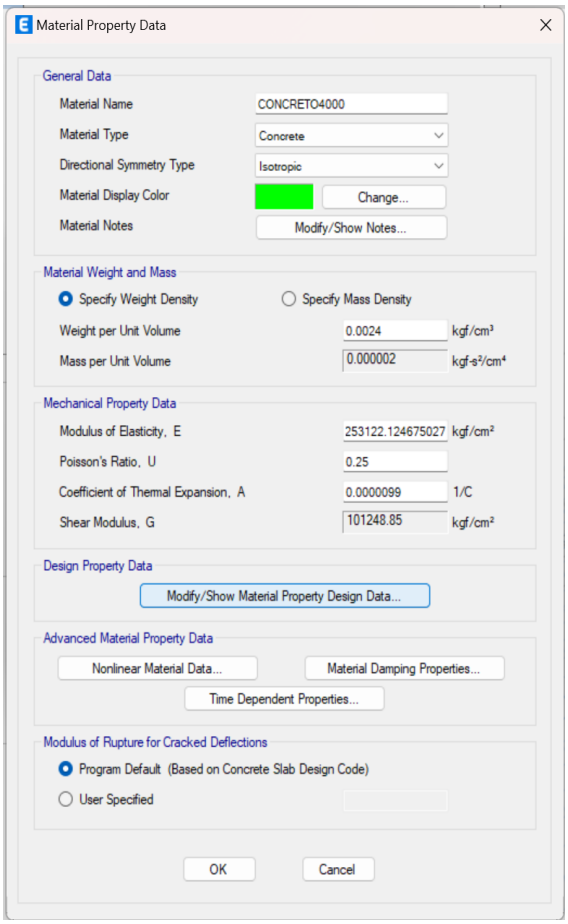
PARA CAMBIAR EL TAMAÑO DE LOS NUMEROS



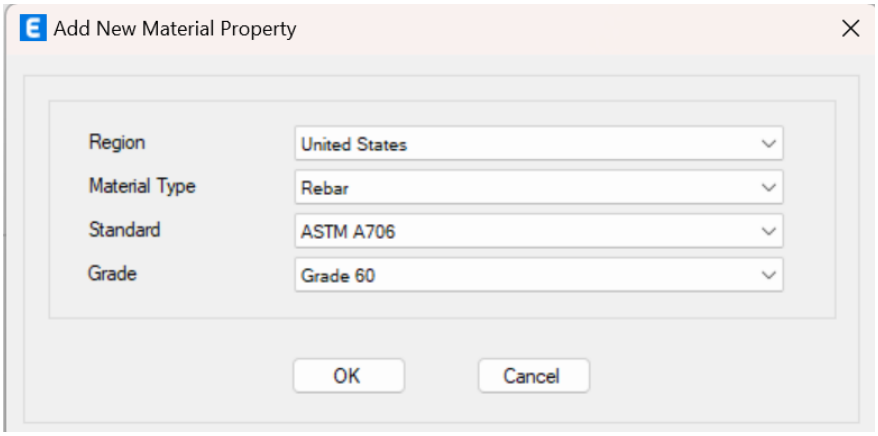
PARA QUE ME APAREZCA EL NOMBRE DEE LAS VIGAS.

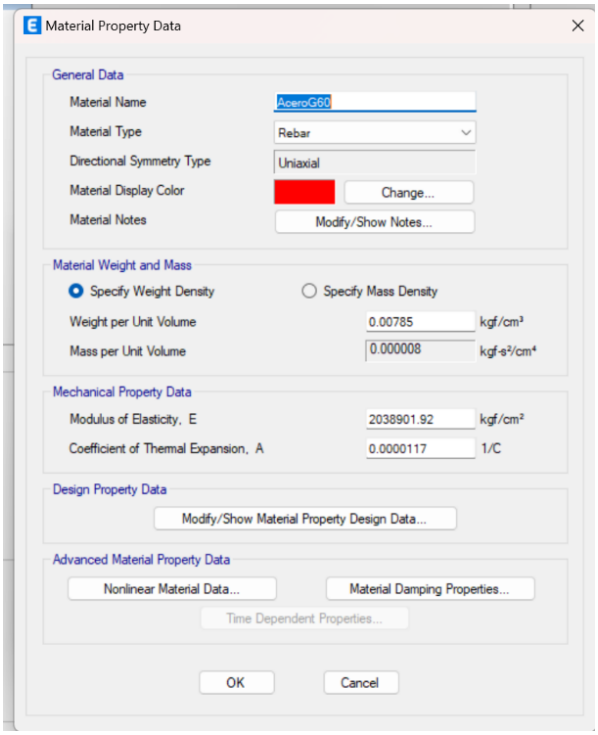
1. **DEFINIR MATERIAL**
   - 1.1. Unnits **cm kgf**
   - 1.2. Define
   - 1.3. Material propiertis
   - 1.4. Add new material.
   - 1.5. Modulus of elasticity, E **15100*sqr(281)**
   - 1.6. Modify/show material property data **281**



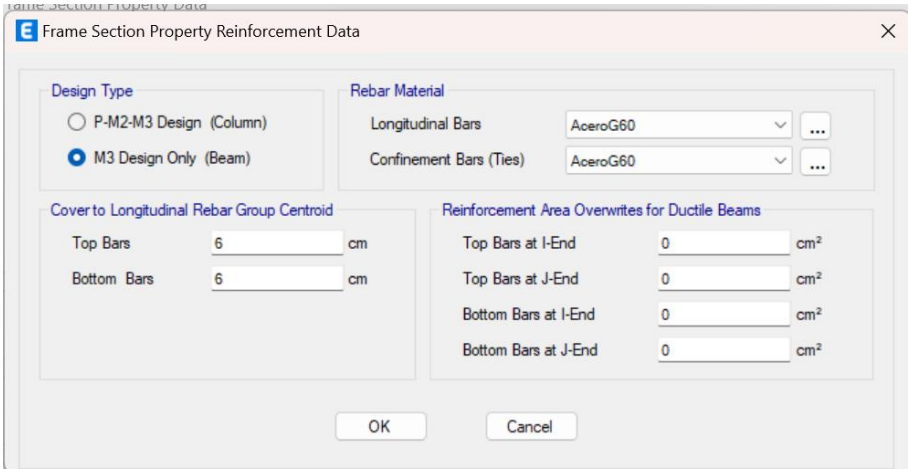


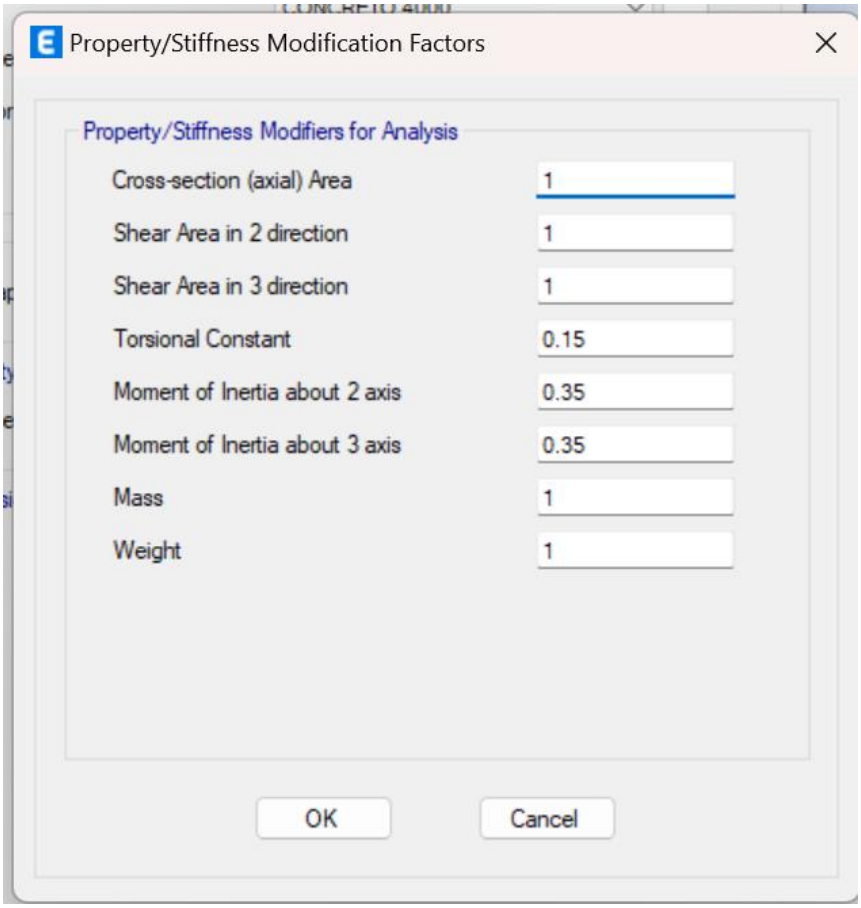
2. **DEFINIR ACERO G60**
   - 2.1. Define
   - 2.2. Material propiertis
   - 2.3. Add new material.
   - 2.4. Cambiarle Nombre **aceroG60**
   - 2.5. Ok (ya no modifica nada)
   - 2.6. ASTM706 solo usa G60
   - 2.7. Astm 615 usa la 40 y 70





3. **CREAR VIGAS**
   - 3.1. Define
   - 3.2. Section propiertis
   - 3.3. Frame section
   - 3.4. Add new propierty
   - 3.5. Section Shape= concrete rectangular
   - 3.6. Cambiar nombre y material elegir el material creado CONCRETO 4000
   - 3.7. Reiforcment > only Beam

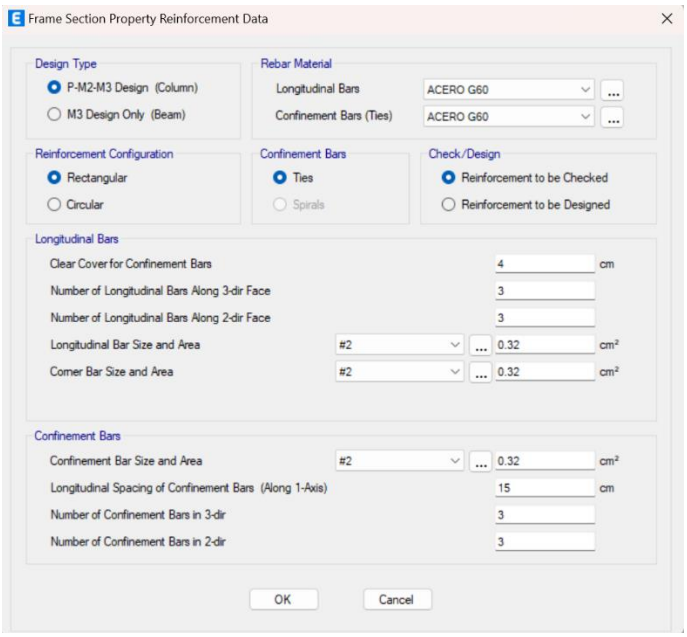


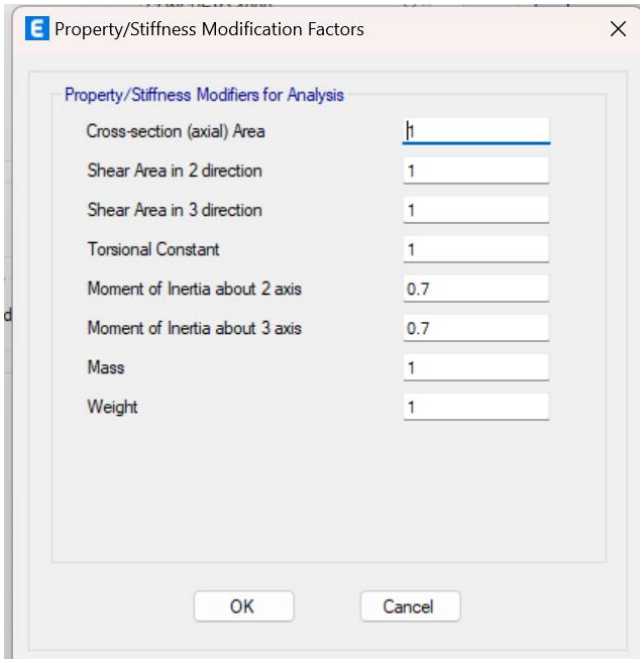


3.8. Delete múltiples propiierty
3.9. Ok>ok
3.10. Para la viga Y se duplica la VX con ADS COPY OF PROPERTY
3.11. Y cambiar Nombre > Ok

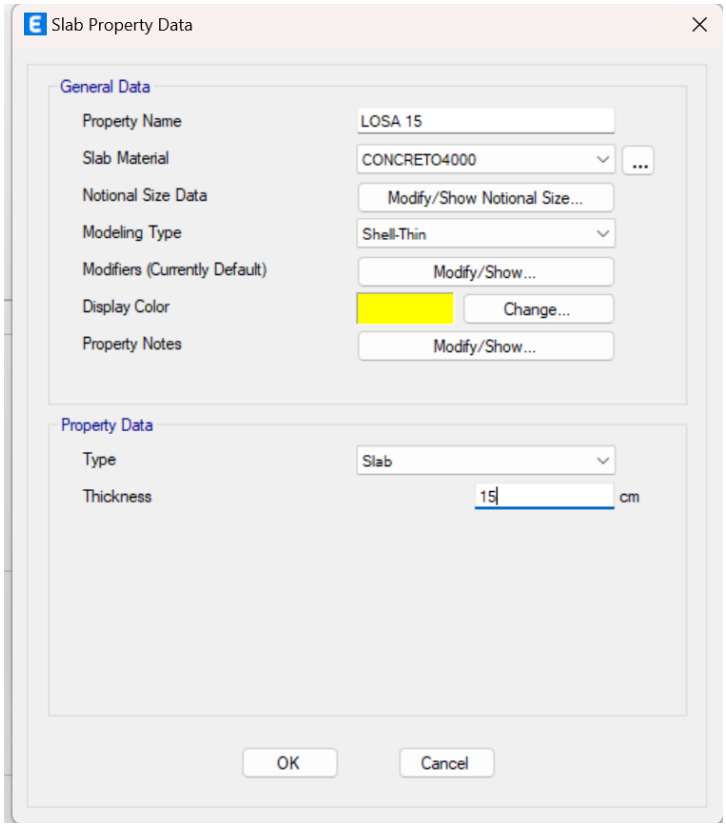
4. **CAMBIAR ACERO PARA COLUMNA.**
    - 4.1. Define
    - 4.2. Section propiertis
    - 4.3. Reiforcing Bar Zises
    - 4.4. Clear all bars
    - 4.5. Add common Bar Set > US CUSTOMARY

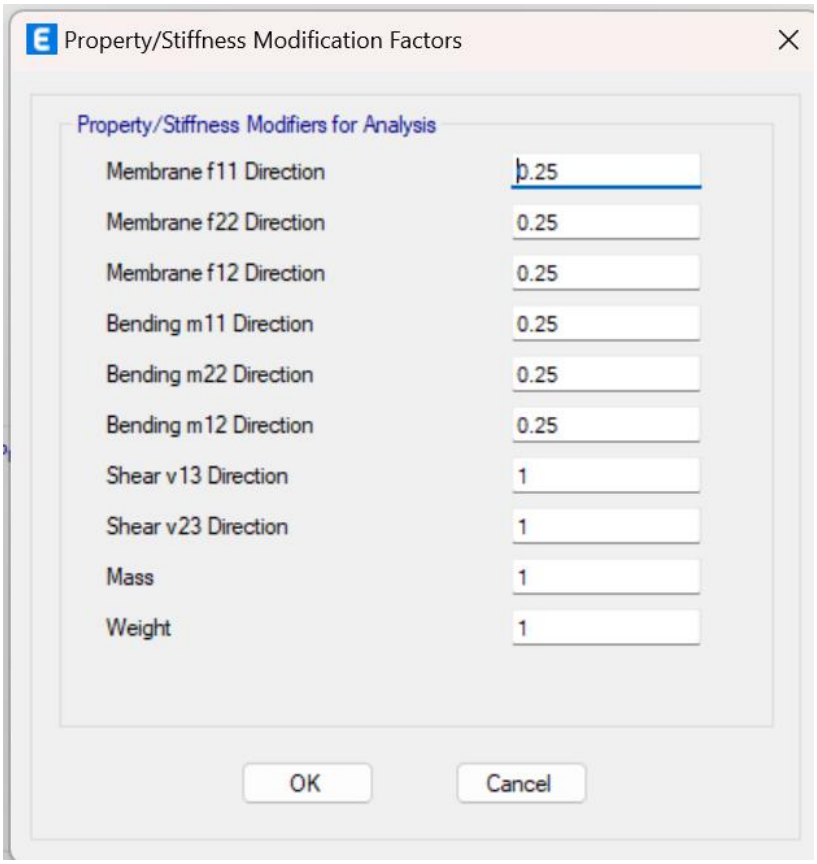
5. **COLUMNA.**
    - 5.1. Define
    - 5.2. Section propiertis
    - 5.3. Frame section
    - 5.4. Add new propierty
    - 5.5. Section Shape= concrete rectangular
    - 5.6. Cambiar nombre y material elegir el material creado CONCRETO 4000
    - 5.7. Reiforcment > Only Colum
    - 5.8. Ok





6. **CREAR LA LOSA.**
    - 6.1. Define
    - 6.2. Section propiertis
    - 6.3. Slab section
    - 6.4. Slab1 > Modify Show Property
    - 6.5. Cambiar nombre
    - 6.6. Slab Material: CONCRETO 4000
    - 6.7. Shell Thin

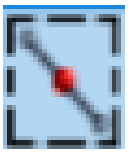

Slab Property Data
General Data
Property Name
LOSA 15
Slab Material
CONCRETO4000
...
Notional Size Data
Modify/Show Notional Size...
Modeling Type
Shell-Thin
Modifiers (Currently Default)
Modify/Show...
Display Color
Change...
Property Notes
Modify/Show...
Property Data
Type
Slab
Thickness
15
cm
OK
Cancel


Property/Stiffness Modification Factors
Property/Stiffness Modifiers for Analysis
Membrane f11 Direction
0.25
Membrane f22 Direction
0.25
Membrane f12 Direction
0.25
Bending m11 Direction
0.25
Bending m22 Direction
0.25
Bending m12 Direction
0.25
Shear v13 Direction
1
Shear v23 Direction
1
Mass
1
Weight
1
OK
Cancel

7. **DIBUJAR COLUMNAS.**

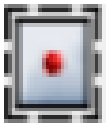
   7.1. Quick draw columns
   7.2. Property > elegir la columna creada
   7.3. Plan View: Story 1
   7.4. Dibujar en cada intersección.

8. **DIBUJAR VIGA**



   8.1. Quick draw beam/column
   8.2. Property > elegir la viga creada (x, y)
   8.3. Plan View: Story 1
   8.4. Dibujar

9. **DIBUJAR LOSAS. (8,9,10)**

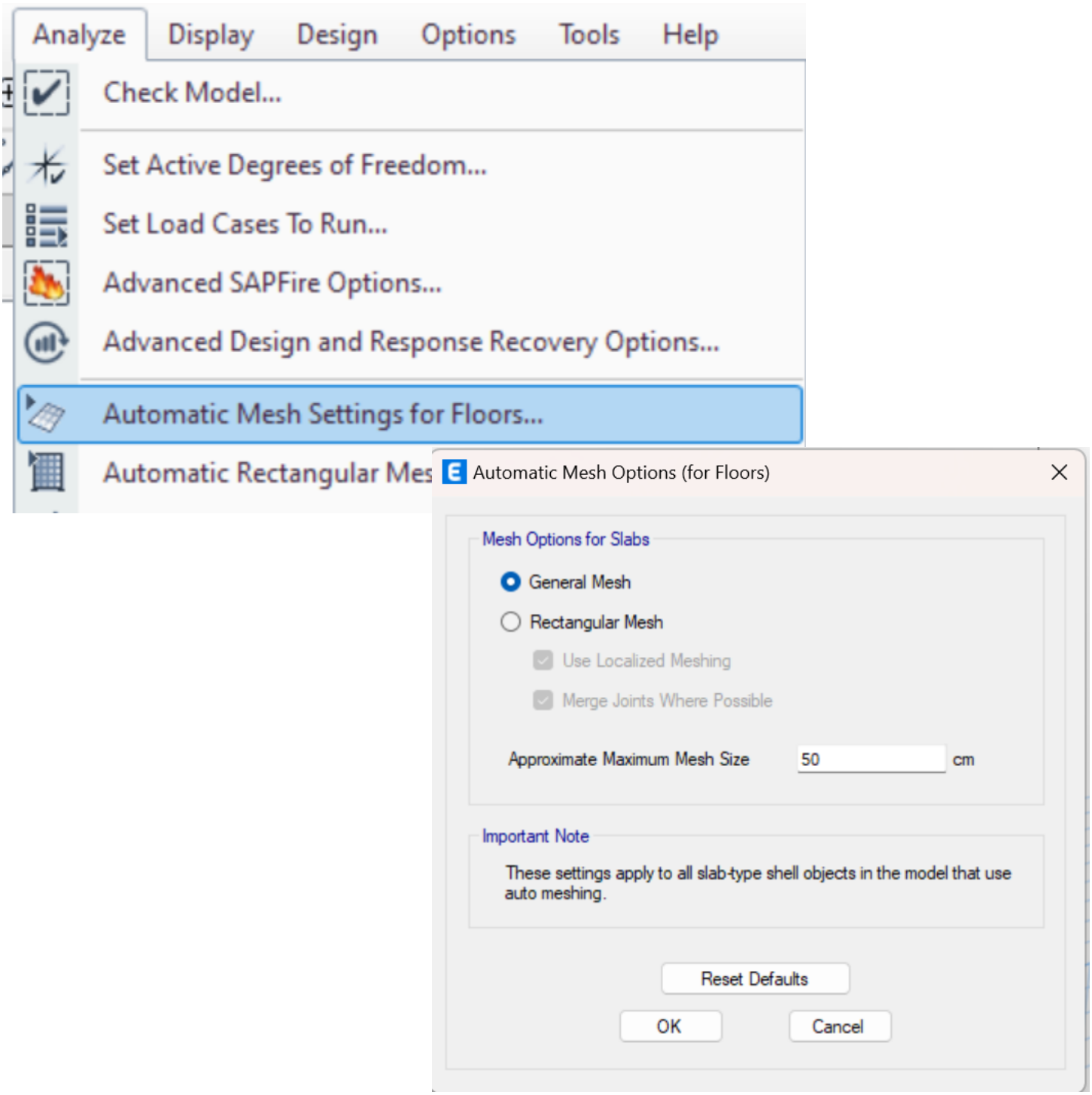


   9.1. Quick draw floor wall
   9.2. Property > Losa creada.

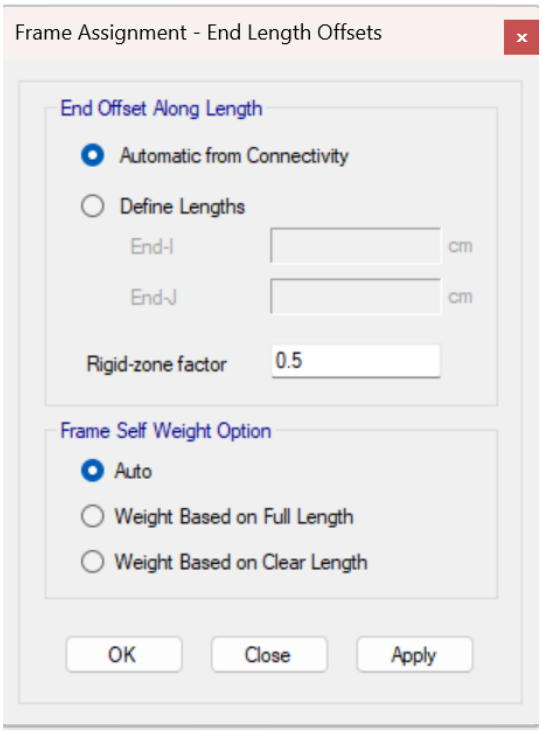
10. **DUPLICAR LO DIBUJADO EN STORY 1 A LOS DEMAS NIVELES.**
   10.1. Seleccionar toda la planta.
   10.2. Edit
   10.3. Replicate.
   10.4. Story.
   10.5. Seleccionar los niveles restantes, donde no se ha dibujado sin la base.
   10.6. Aply > Ok.

11. **EMPOTRAR LA BASE.**
   11.1. Ir a la base > seleccionar toda la base.
   11.2. Assign
   11.3. Joints
   11.4. Restraints.
   11.5. Aply > Ok.

12. **PONER MALLADO EN LOSAS.**
   12.1. SELECT
   12.2. Select
   12.3. Properties > Slab Section > select
   12.4. Show select object only
   12.5. ANALYZE

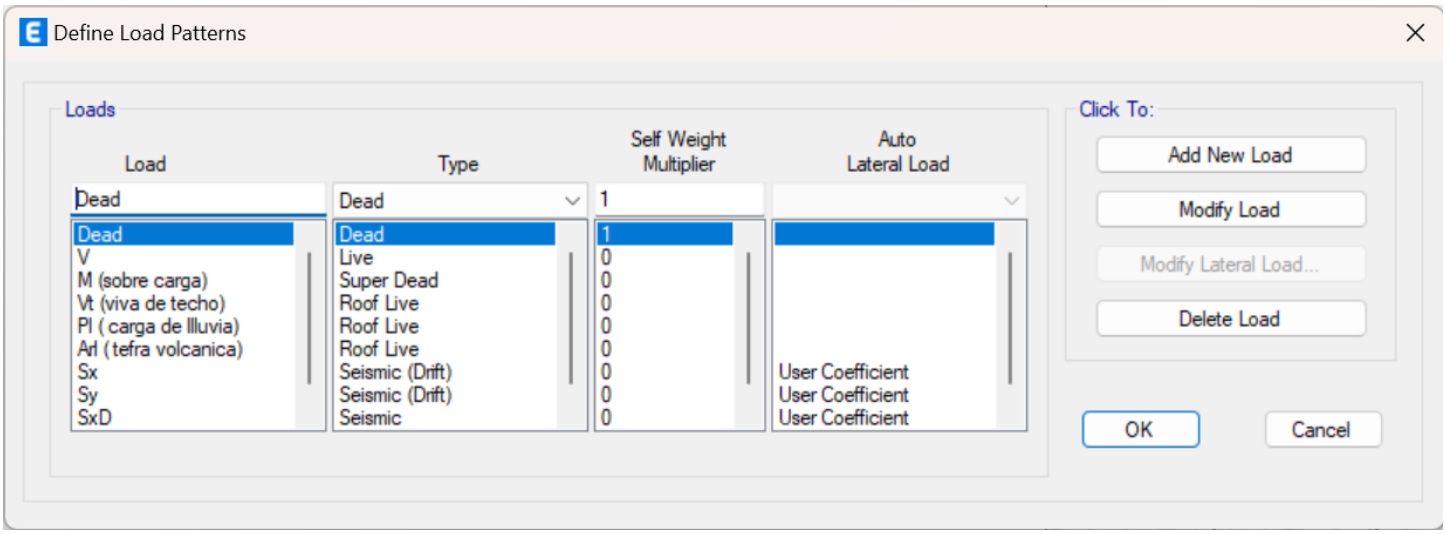

Analyze
Display
Design
Options
Tools
Help
Check Model...
Set Active Degrees of Freedom...
Set Load Cases To Run...
Advanced SAPFire Options...
Advanced Design and Response Recovery Options...
Automatic Mesh Settings for Floors...
Automatic Rectangular Mes
Automatic Mesh Options (for Floors)
Mesh Options for Slabs
General Mesh
Rectangular Mesh
Use Localized Meshing
Merge Joints Where Possible
Approximate Maximum Mesh Size
50
cm
Important Note
These settings apply to all slab-type shell objects in the model that use auto meshing.
Reset Defaults
OK
Cancel

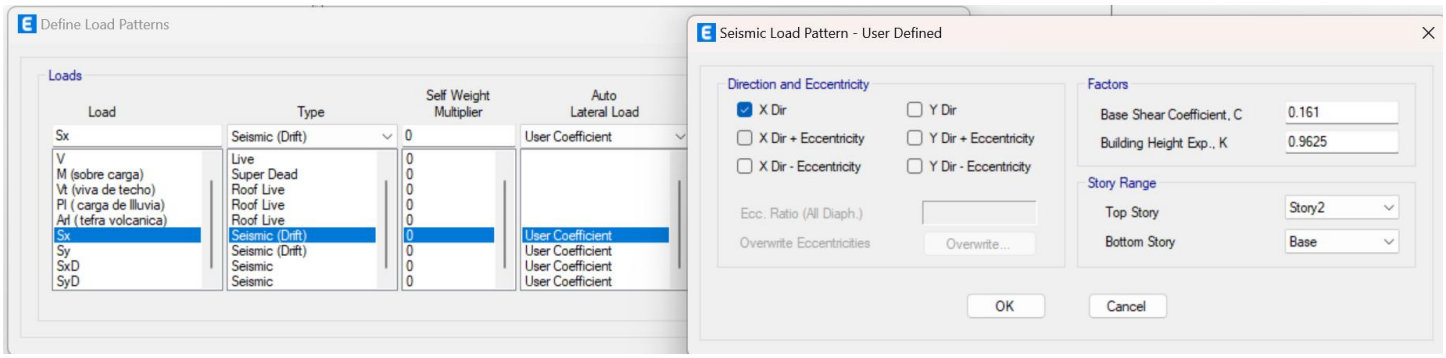
13. **ASIGNAR SECCIONES RIGIDAS A LOS NODOS**
    - 13.1. Seleccionar el 3D
    - 13.2. ASSIGN
    - 13.3. Frame > End Length Offsets > Factor rigid zone = 0.5
    - 13.4. Aply.

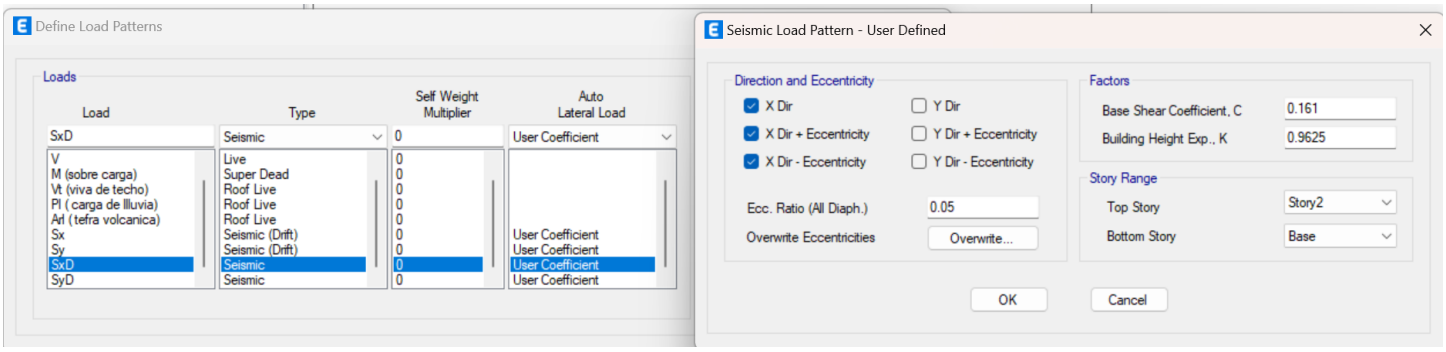


14. **DEFINIR DIAFRAGMA RIGIDO POR NIVEL**
    - 14.1. DEFINE
    - 14.2. Diafragmas
    - 14.3. Add new diafragma > Rigid > crear los que sean necesarios.
    - 14.4. ASSIGN
    - 14.5. SHELL
    - 14.6. Diafragma
    - 14.7. Seleccionar nivel 1 completo y aplicar D1
    - 14.8. Asignarle D2 a Story 2…

15. **DEFINIR PATRONES DE CARGA**
    - 15.1. DEFINE
    - 15.2. Load paterns

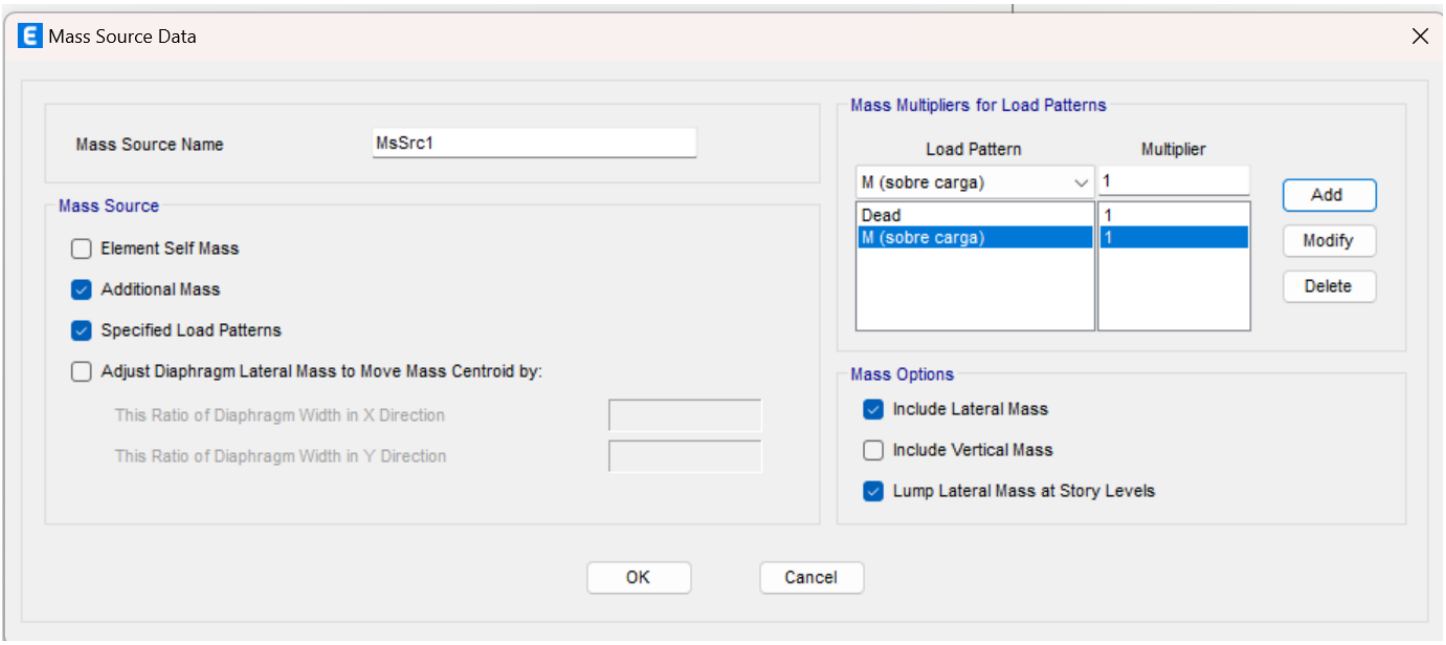






16. **DEFINIR CASOS DE CARGAS.**
    - 16.1. DEFINE
    - 16.2. Mass Source > modificar > cambiar nombre masa sísmica
    - 16.3. Specifed Load Paterns

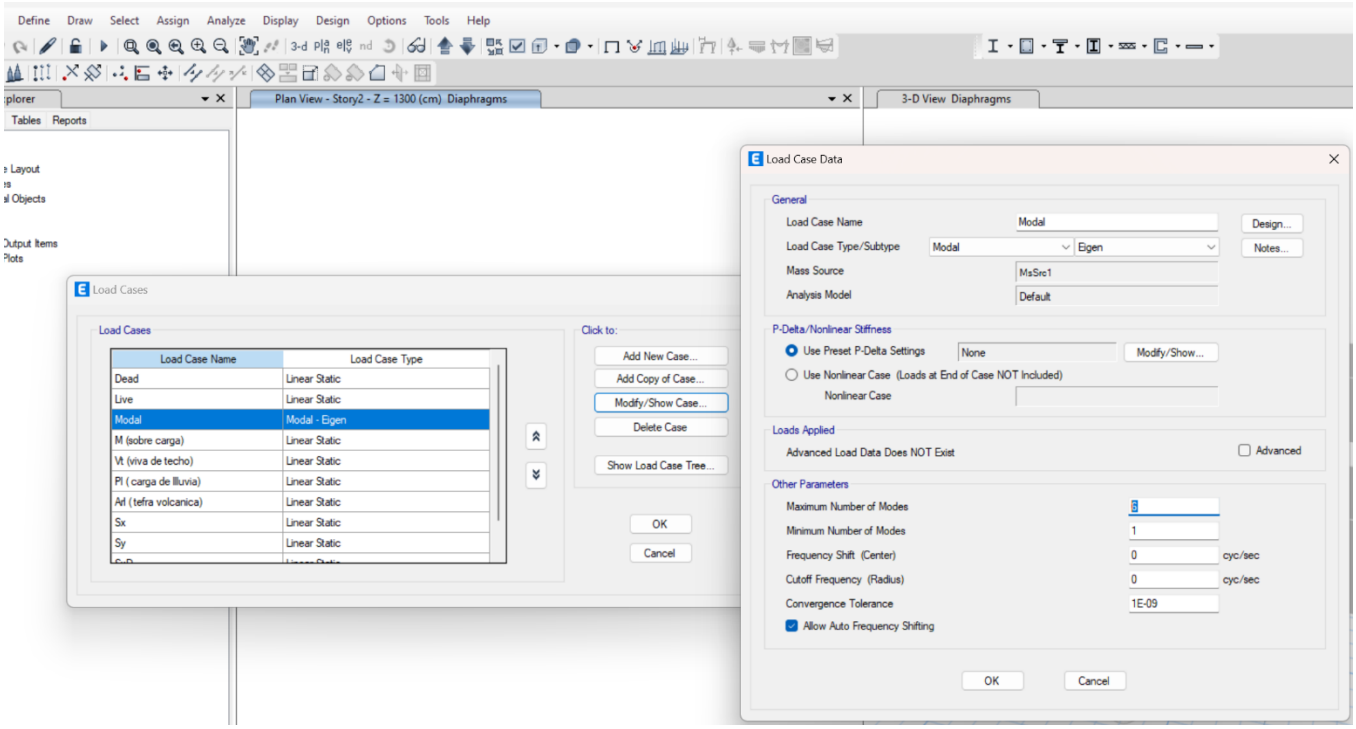
NOTA: SOLO SI LA CV > 500kg/m² se considera el 0.25 de la carga viva



16.4. Ok > Ok

17. CAMBIARLE EL VALOR AL CASO NODAL.

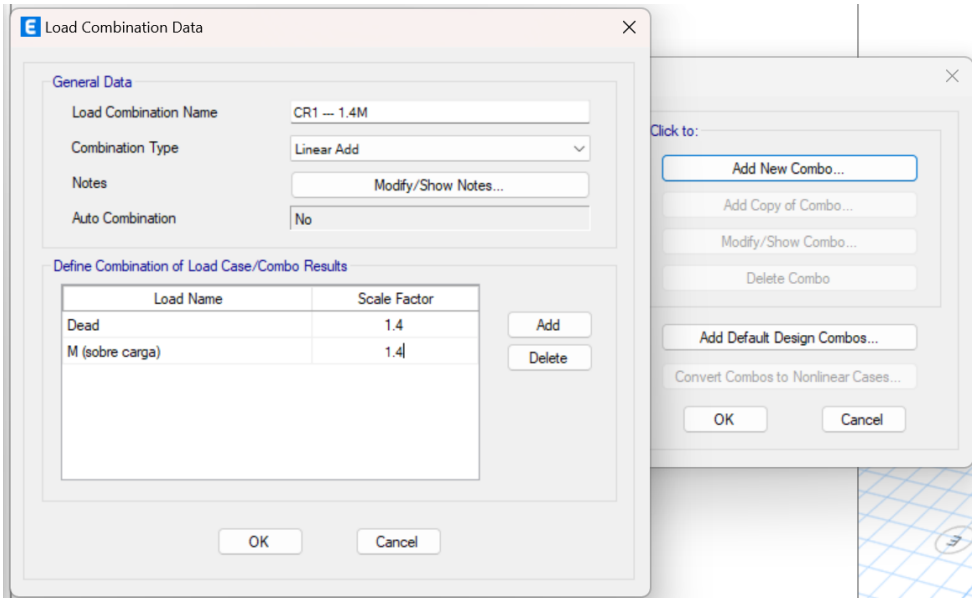
17.1. DEFINE.
17.2. Load cases.

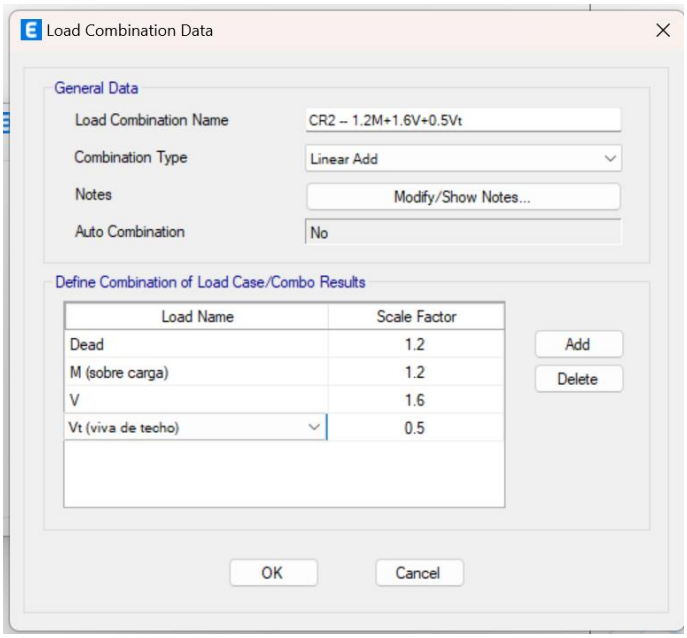


NOTA: SE MODIFICA EL VALOR = 3XCANT DE NIVELES > **3X2niveles = 6**

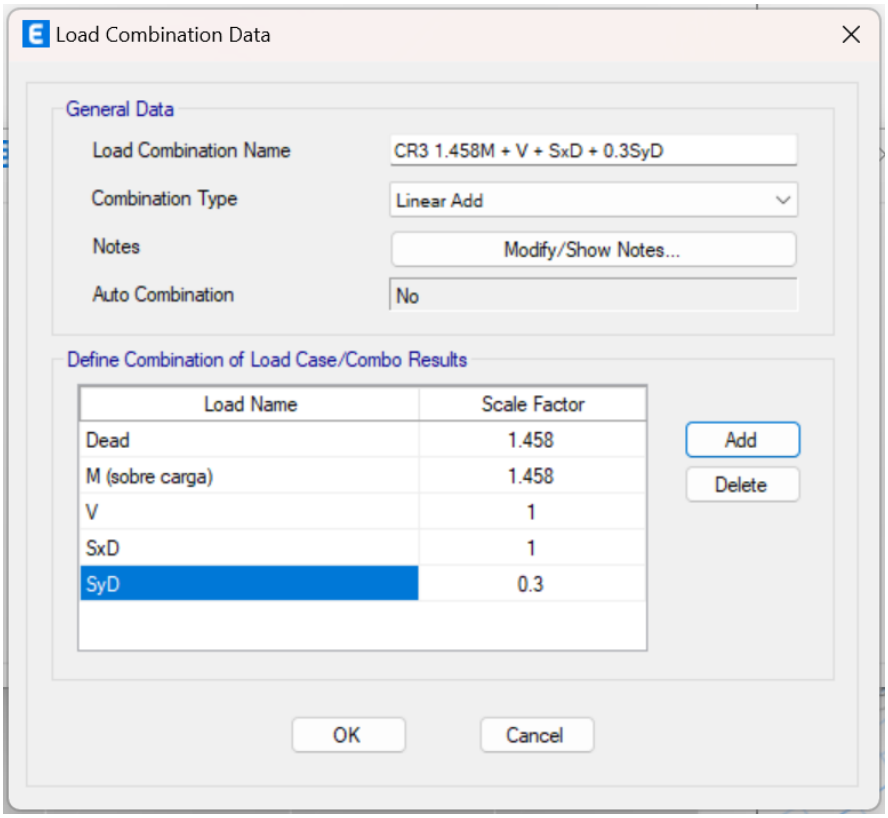
18. **COMBINACIONES DE CARGA.**

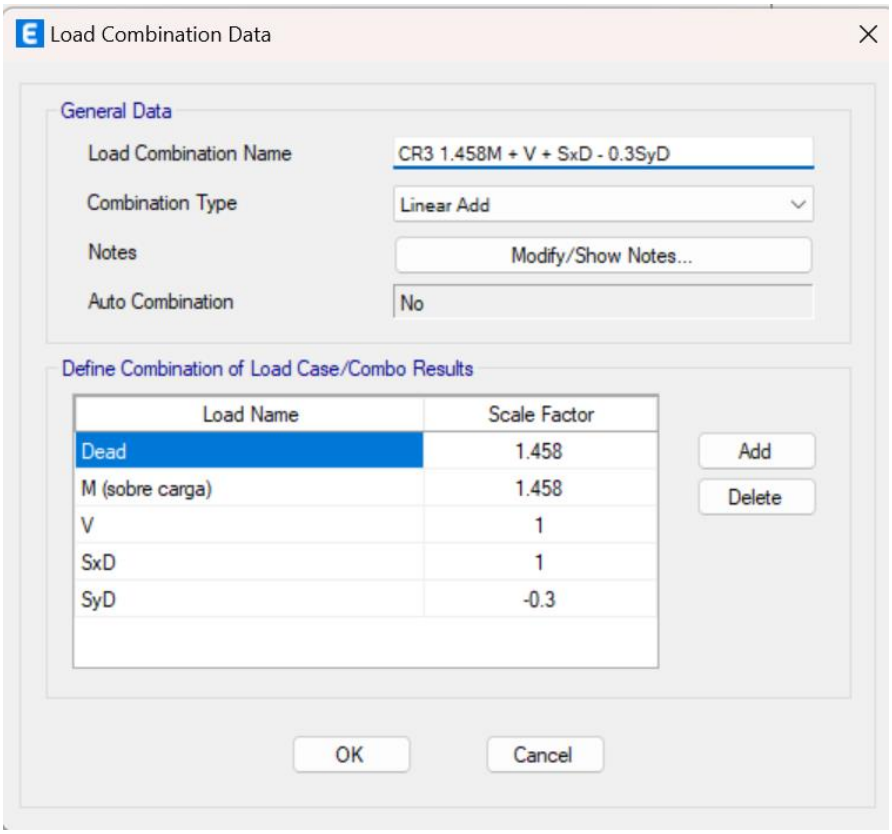
18.1. DEFINE.
18.2. Load combinations
18.3. Add combo > crear las 4 gravitacionales.
18.4. El resto crearlo por defecto.



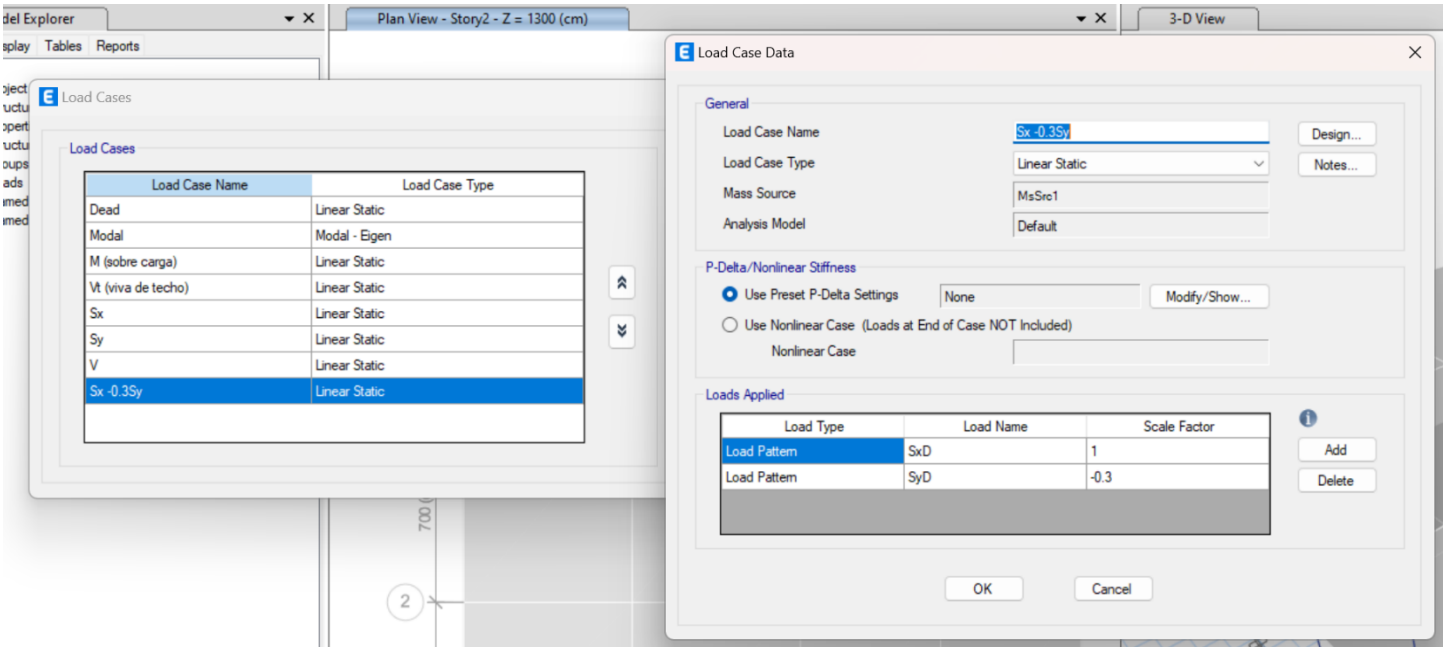


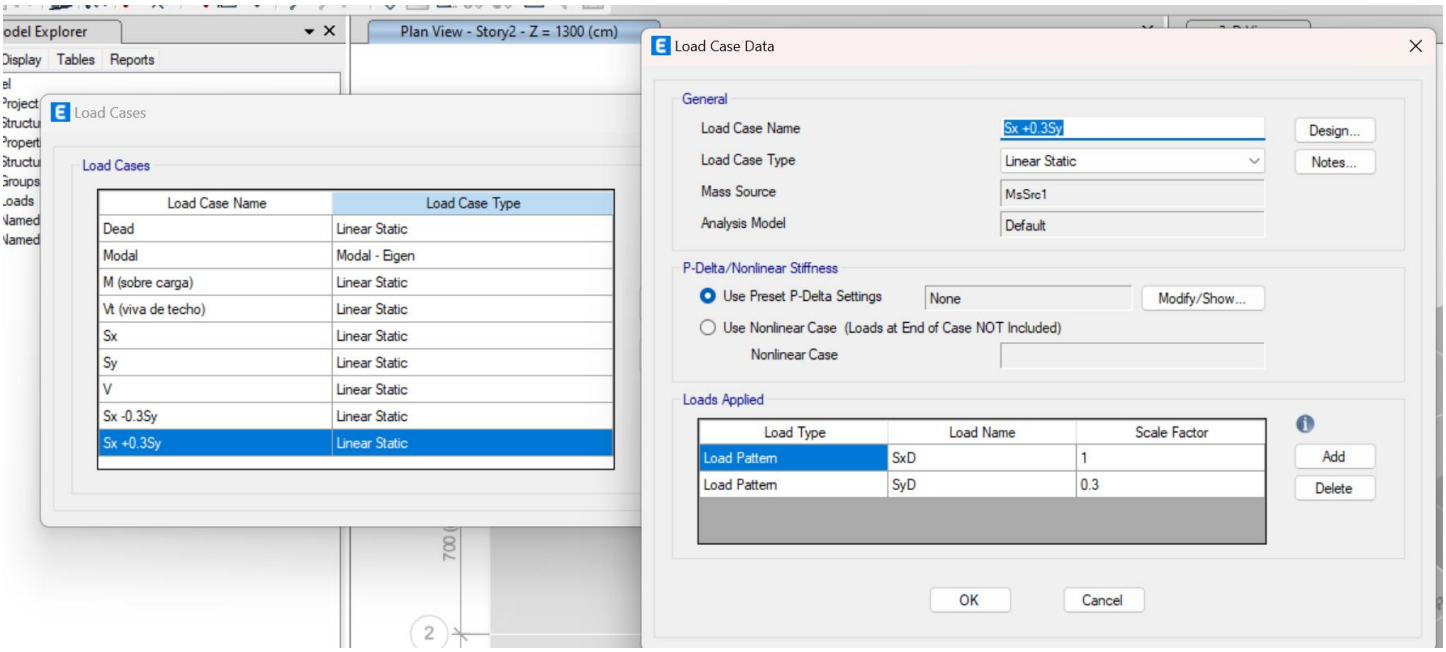
1.2+Svd

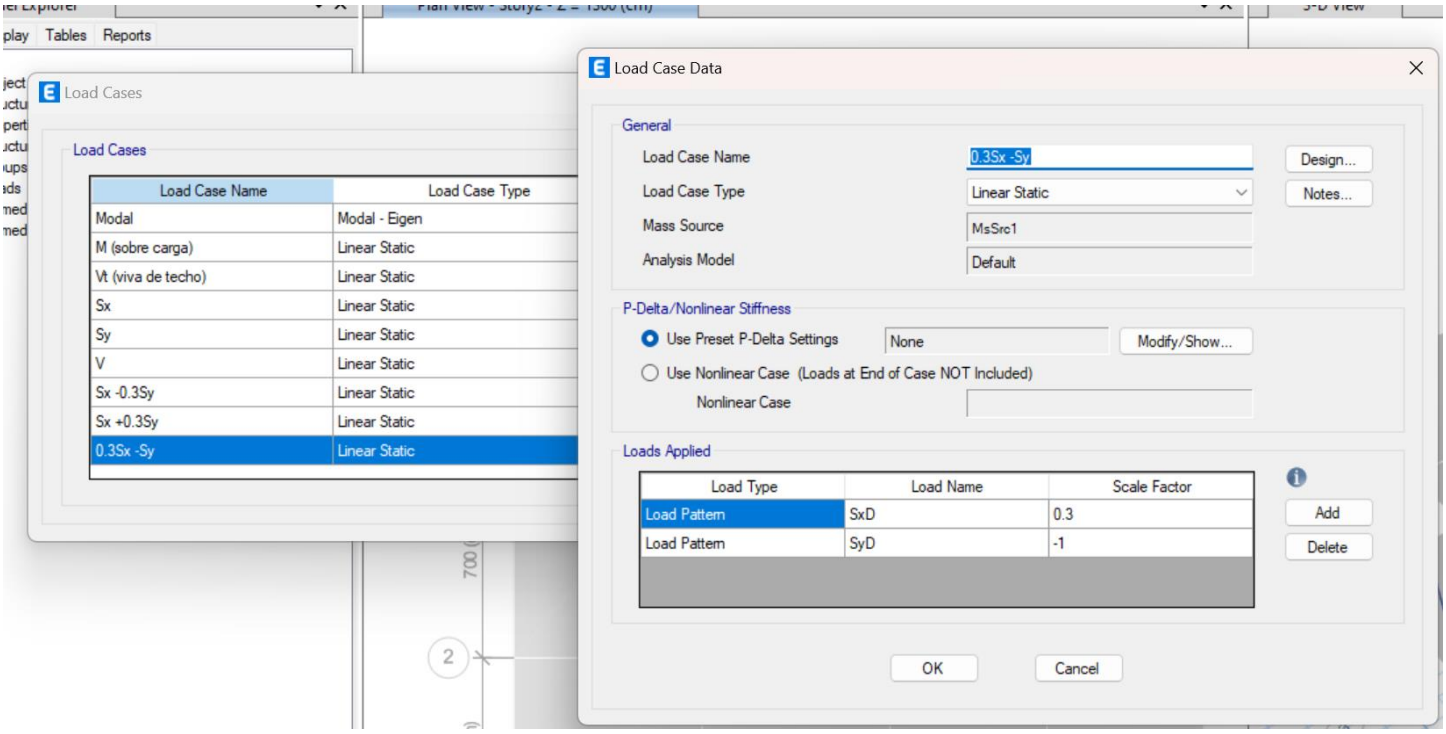

Load Combination Data
General Data
Load Combination Name
CR3 1.458M + V + SxD + 0.3SyD
Combination Type
Linear Add
Notes
Modify/Show Notes...
Auto Combination
No
Define Combination of Load Case/Combo Results
Load Name
Scale Factor
Dead
1.458
M (sobre carga)
1.458
V
1
SxD
1
SyD
0.3
Add
Delete
OK
Cancel

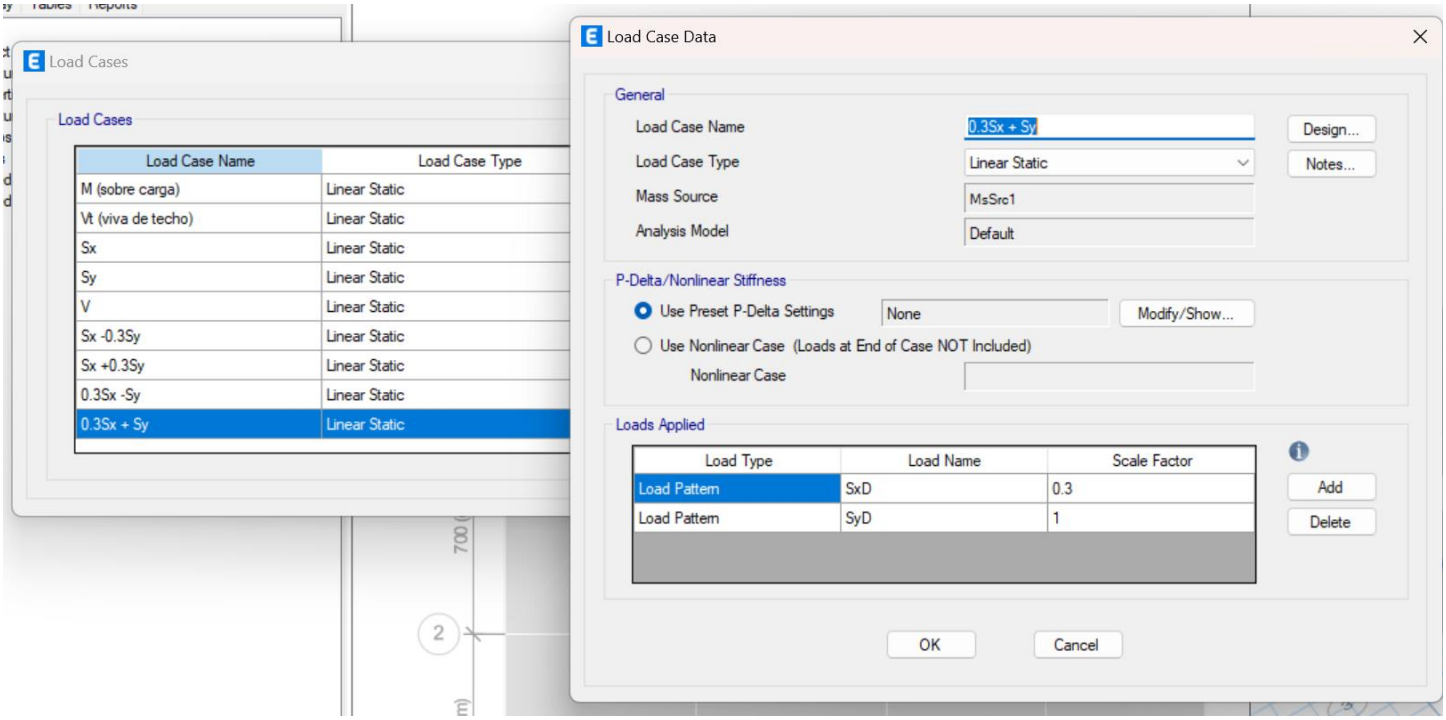

Load Combination Data
General Data
Load Combination Name
CR3 1.458M + V + SxD - 0.3SyD
Combination Type
Linear Add
Notes
Modify/Show Notes...
Auto Combination
No
Define Combination of Load Case/Combo Results
Load Name
Scale Factor
Dead
1.458
M (sobre carga)
1.458
V
1
SxD
1
SyD
-0.3
Add
Delete
OK
Cancel

19. **PARA CREAR LAS COMBINACIONES AUTOMATICAMENTE.**
    19.1. DEFINE.
    19.2. Load Cases
    19.3. Eliminar PI y Ar porque el valor de Vt es el valor mayor de los 3
    19.4. Eliminar los SxD y SyD
    19.5. Crear un nuevo caso de carga





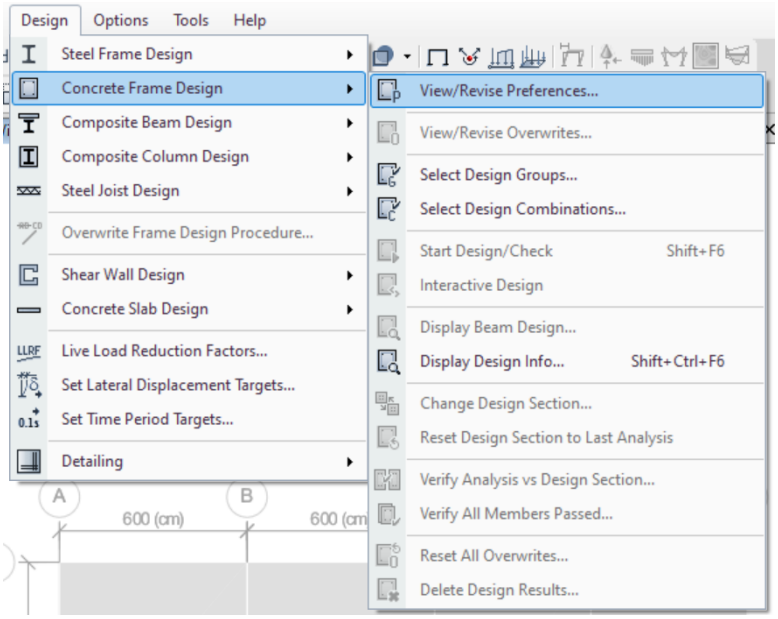


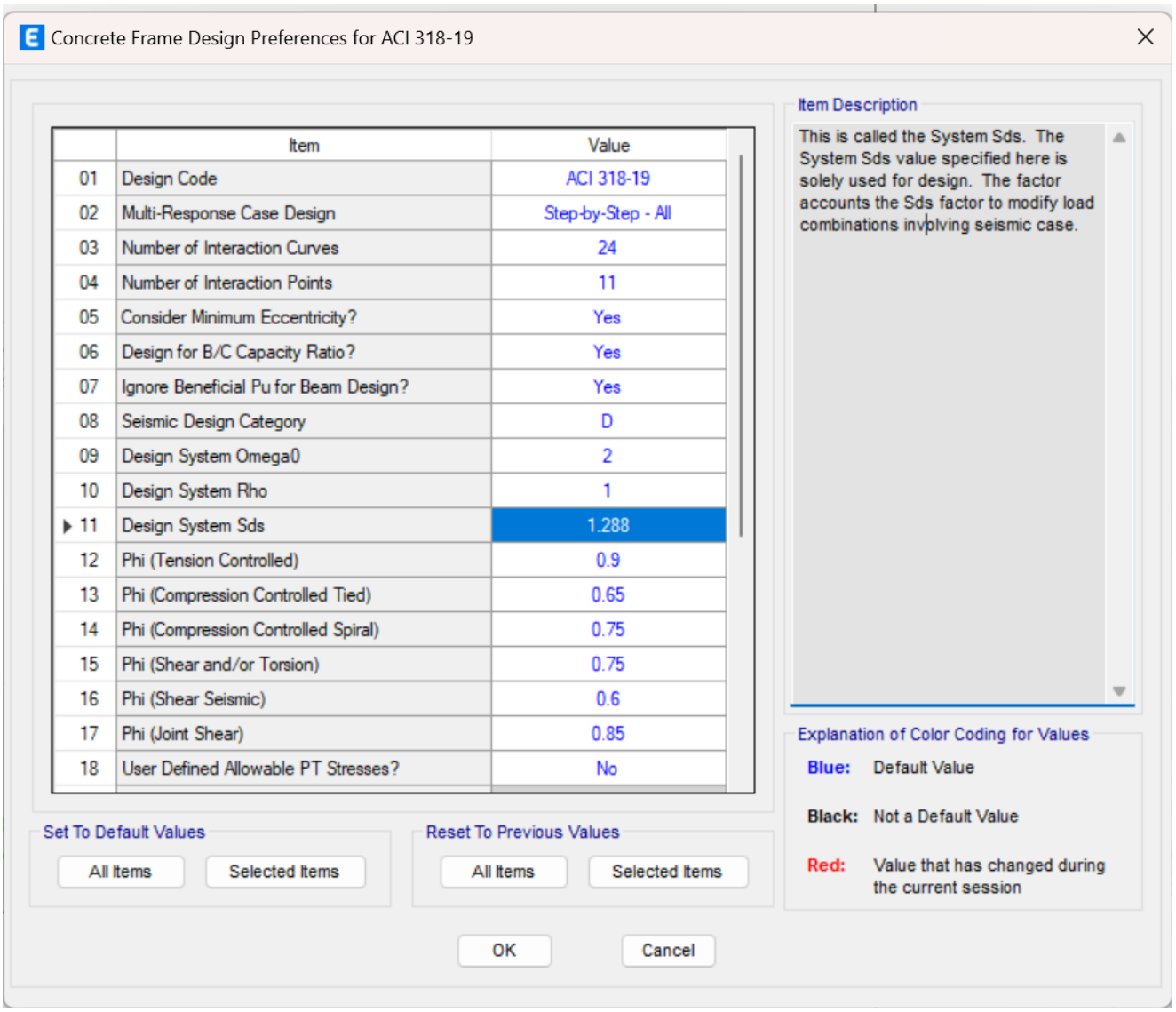


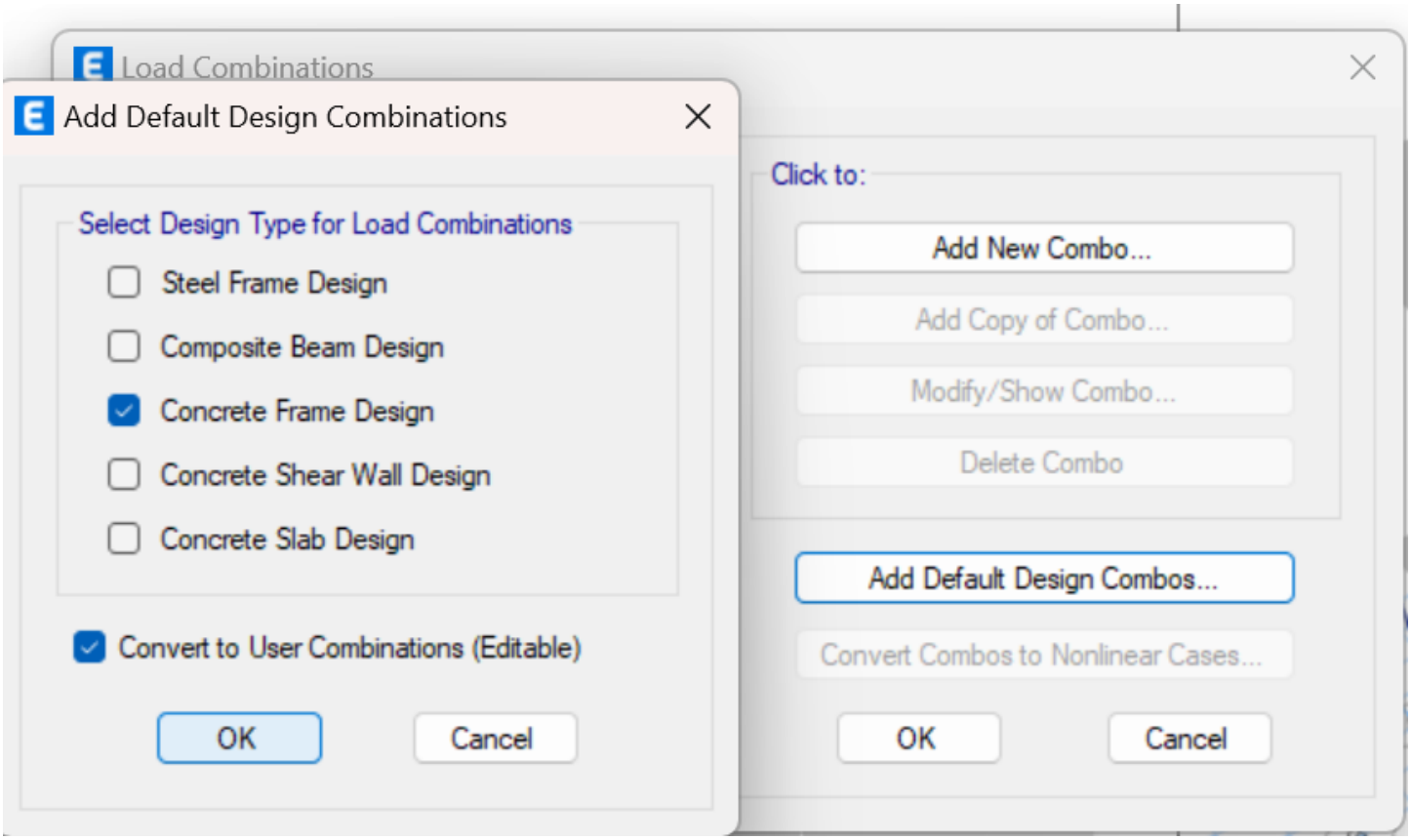
19.6. **DESIGN.**

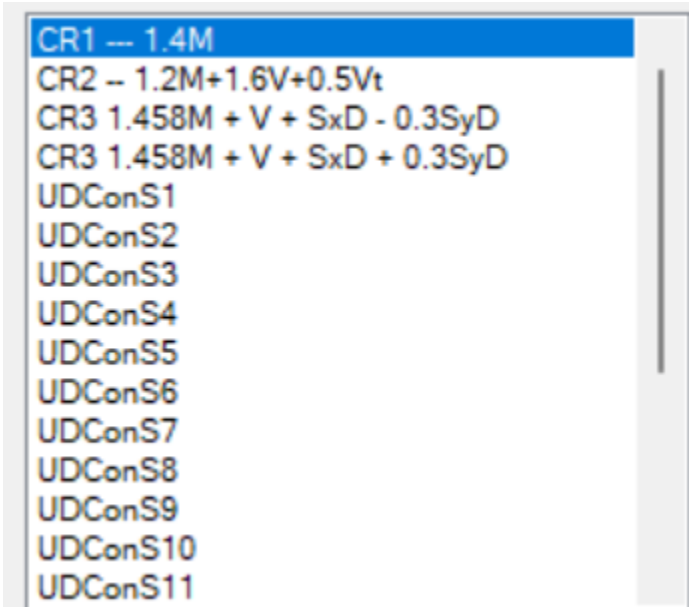
19.7. CONCRETE FRAME DESIGN

19.8. Cambiar el DESIGN SISTEM SDS POR EL Scd DEL ESPECTRO.


Design
Options
Tools
Help
Steel Frame Design
Concrete Frame Design
Composite Beam Design
Composite Column Design
Steel Joist Design
Overwrite Frame Design Procedure...
Shear Wall Design
Concrete Slab Design
Live Load Reduction Factors...
Set Lateral Displacement Targets...
Set Time Period Targets...
Detailing
View/Revise Preferences...
View/Revise Overwrites...
Select Design Groups...
Select Design Combinations...
Start Design/Check Shift+F6
Interactive Design
Display Beam Design...
Display Design Info... Shift+Ctrl+F6
Change Design Section...
Reset Design Section to Last Analysis
Verify Analysis vs Design Section...
Verify All Members Passed...
Reset All Overwrites...
Delete Design Results...
A
B
600 (cm)
600 (cm

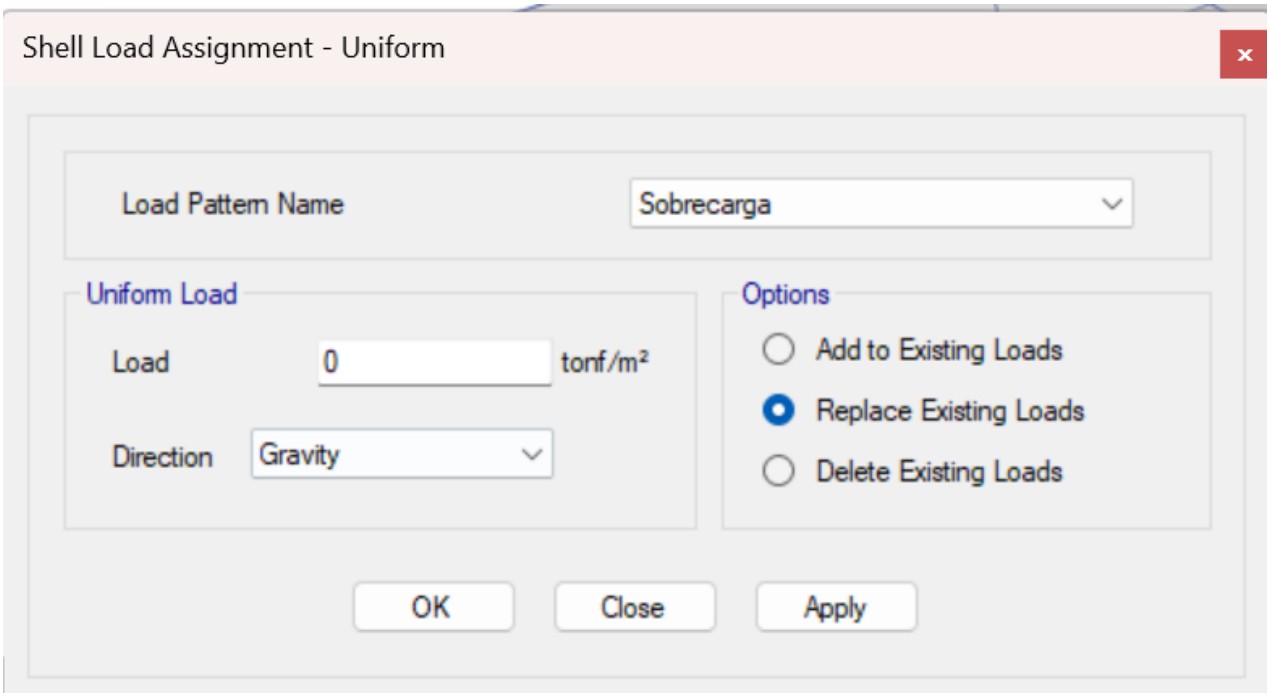

Concrete Frame Design Preferences for ACI 318-19
Item
Value
01 Design Code ACI 318-19
02 Multi-Response Case Design Step-by-Step - All
03 Number of Interaction Curves 24
04 Number of Interaction Points 11
05 Consider Minimum Eccentricity? Yes
06 Design for B/C Capacity Ratio? Yes
07 Ignore Beneficial Pu for Beam Design? Yes
08 Seismic Design Category D
09 Design System Omega0 2
10 Design System Rho 1
11 Design System Sds 1.288
12 Phi (Tension Controlled) 0.9
13 Phi (Compression Controlled Tied) 0.65
14 Phi (Compression Controlled Spiral) 0.75
15 Phi (Shear and/or Torsion) 0.75
16 Phi (Shear Seismic) 0.6
17 Phi (Joint Shear) 0.85
18 User Defined Allowable PT Stresses? No
Item Description
This is called the System Sds. The System Sds value specified here is solely used for design. The factor accounts the Sds factor to modify load combinations involving seismic case.
Explanation of Color Coding for Values
Blue: Default Value
Black: Not a Default Value
Red: Value that has changed during the current session
Set To Default Values
All Items
Selected Items
Reset To Previous Values
All Items
Selected Items
OK
Cancel



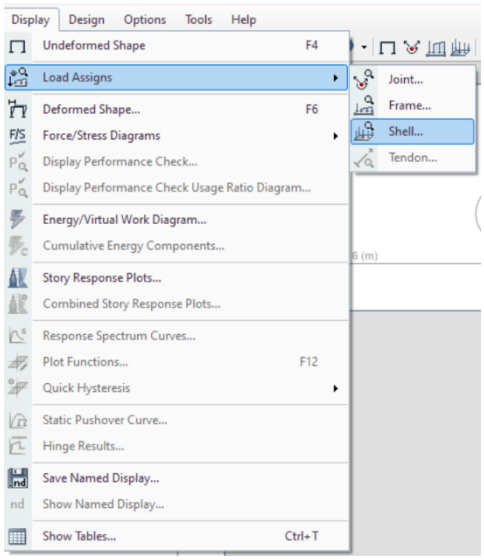


20. **ASIGNAR SOBRE CARGAS ENTREPISO Y TECHO.kg/m²/gravitacional.**

20.1. ASSIGN
20.2. Shell Loads > Unniform
20.3. Seleccionar solo losas story 1 y 2
20.4. Ultimo nivel cambia



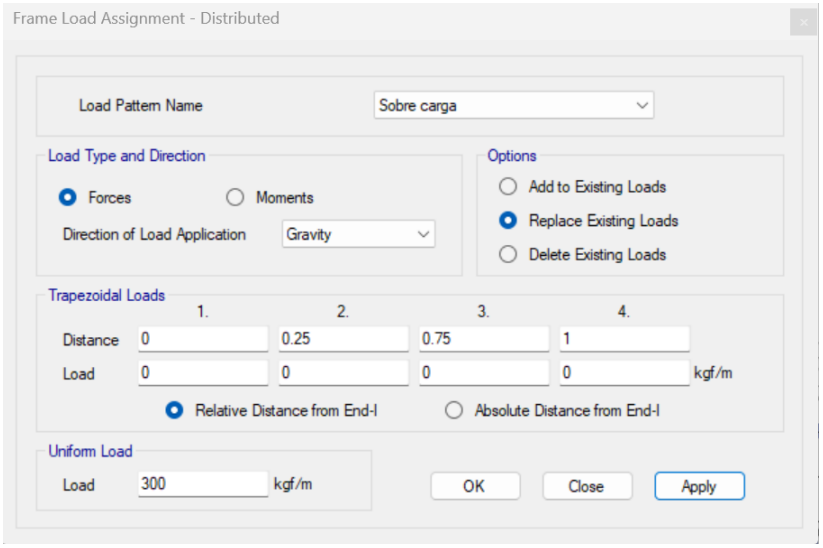
21. **ASIGNAR CARGA VIVA Y (VIVA DE TECHO PARA EL ULTIMO NIVEL).**
    - 21.1. ASSIGN
    - 21.2. Shell Loads > Unniform
    - 21.3. Seleccionar carga viva/viva de techo
    - 21.4. Seleccionar solo losas story 1 y 2
    - 21.5. Ultimo nivel cambia a viva de techo.

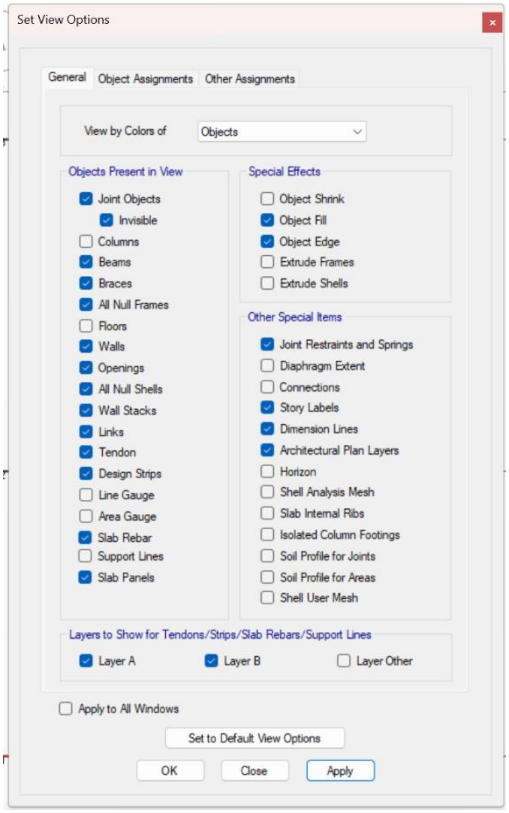


SI QUIERO QUE ME APAREZCAN LAS CARGAS EN LAS LOSAS

22. **ASIGNAR SOBRECARGA A LAS VIGAS.**
    - 22.1. Seleccionar solo vigas
    - 22.2. ASSIGN

22.3. Frame Loads > Distribuide
22.4. Seleccionar sobre carga



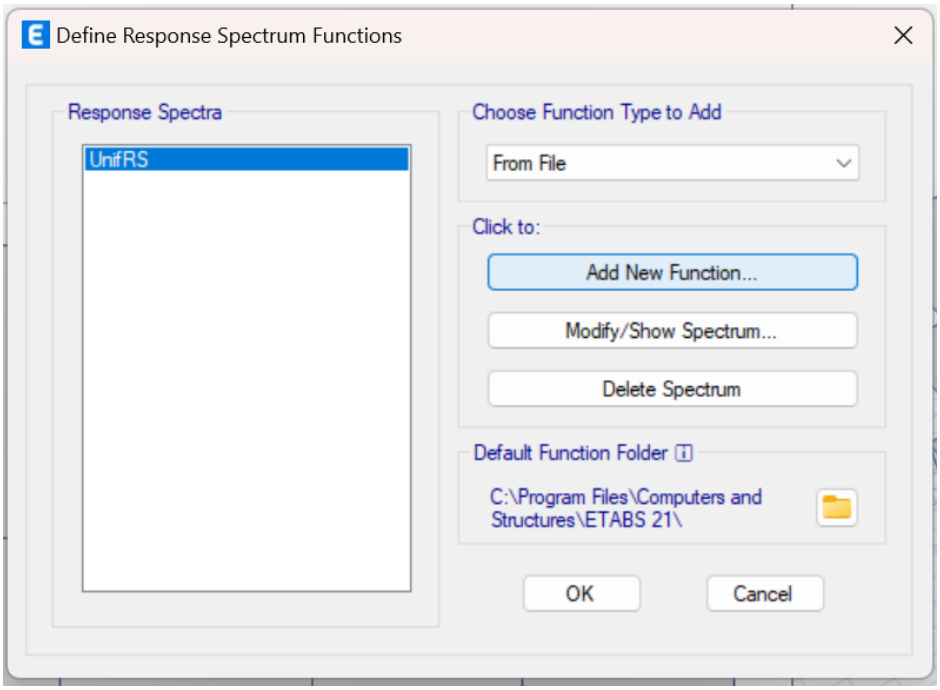


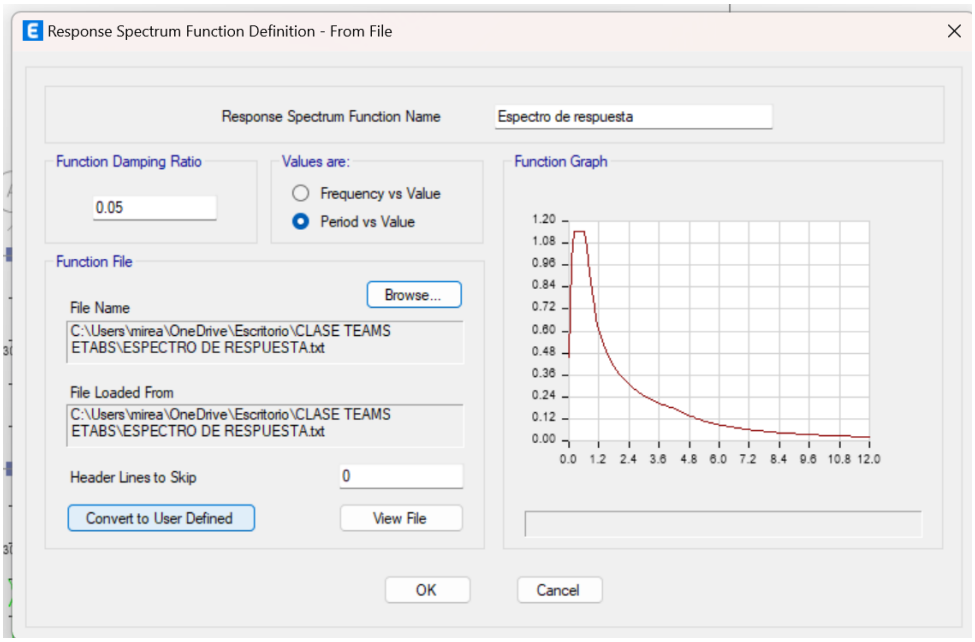
23. TRABAJAR LA CARGA SISMICA.

23.1. Trabajar el espectro de respuesta siguiendo las normas.

24. CARGAR EL ESPECTRO DE RESPUESTA AL ETABS

24.1. DEFINE
24.2. Functions
24.3. Response Spectrum.
24.4. Fom file > Add New Function > Browse





24.5. Convert to user defined > Ok > Ok

25. **PARA CORRER EL PROGRAMA**

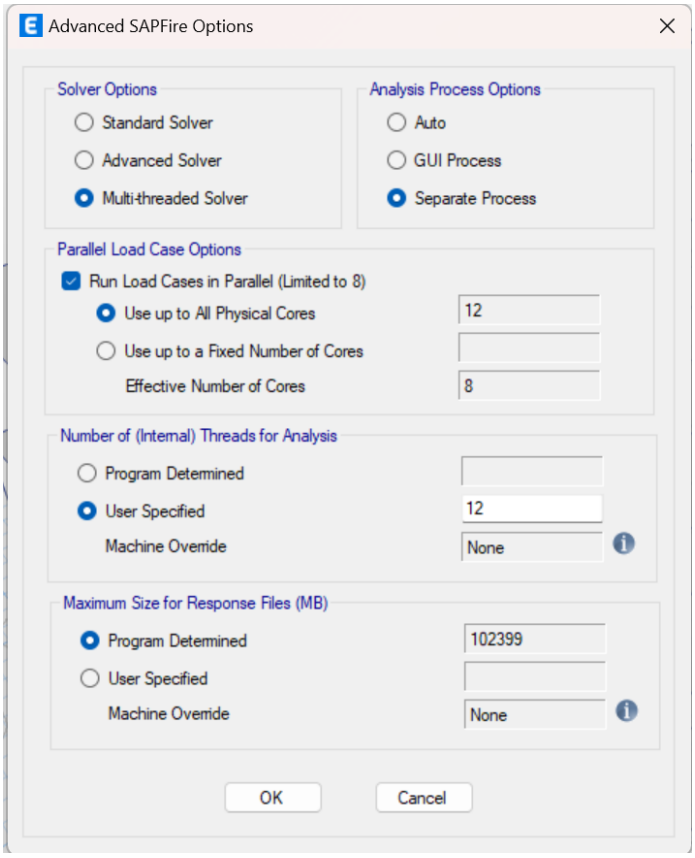
25.1. ANALIZE

25.2. Check model > seleccionar todos.

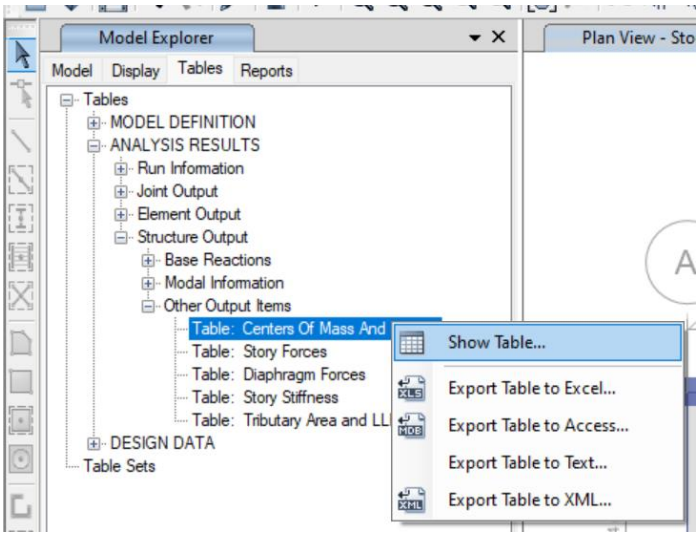
25.3. Advanced SAPFire Option (4ta opción) > para que corra mas rápido.

25.4. Seat load cases to run

25.5. Ok



26. **PARA OBTENER DATOS DE CENTRO DE MASA Y RIGIDEZ.**
    26.1. **MODEL EXPLORER**
    26.2. TABLA
    26.3. ANALISIS RESULTADOS
    26.4. STRUCTURE OUTPUT
    26.5. Other Output Items > Center of Mass and Rigidity > Show table.



27. **CORTANTE BASAL Y CORTANTE POR PISO.**
    27.1. **MODEL EXPLORER**
    27.2. TABLA
    27.3. ANALISIS RESULTADOS
    27.4. STRUCTURE OUTPUT
    27.5. BASE REACTION > base reaction > SHOW TABLE
    27.6. **OUTPUT CASE: SISMOESTATICO**

Output Case	Case Type	Step Type	Step Number	Step Label	FX kgf	FY kgf	FZ kgf	MX kgf-m	MY kgf-m
Sismo Estatico	LinStatic	Step By Step	1		-50930.19	0	0	0	-360339.24
Sismo Estatico	LinStatic	Step By Step	2		0	-50930.19	0	360339.24	0
Sismo Estatico	LinStatic	Step By Step	3		-50930.19	0	0	0	-360339.24
Sismo Estatico	LinStatic	Step By Step	4		0	-50930.19	0	360339.24	0
Sismo Estatico	LinStatic	Step By Step	5		-50930.19	0	0	0	-360339.24
Sismo Estatico	LinStatic	Step By Step	6		0	-50930.19	0	360339.24	0

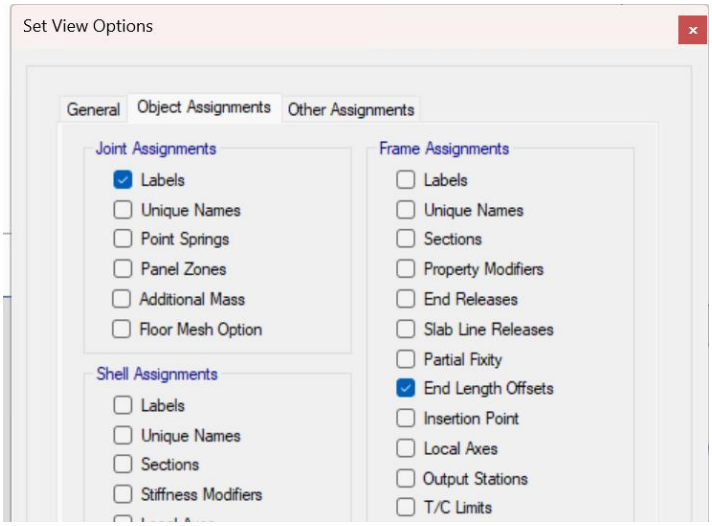
28. **FUERZA POR PISO ACUMULADAS**
    28.1. **MODEL EXPLORER**
    28.2. TABLA
    28.3. ANALISIS RESULTADOS
    28.4. STRUCTURE OUTPUT
    28.5. Other Output Ítems > Story Forces > Show Table.
    28.6. UOTPUT CASE: sismoestatico

28.7. Step Number: x=1,y=2

	Story	Output Case	Case Type	Step Type	Step Number	Step Label	Location	P tonf	VX tonf	VY tonf
▶	Story3	Sismo Estatico	LinStatic	Step By Step	1		Top	0	-26.4221	0
	Story3	Sismo Estatico	LinStatic	Step By Step	1		Bottom	0	-26.4221	0
	Story2	Sismo Estatico	LinStatic	Step By Step	1		Top	0	-42.7608	0
	Story2	Sismo Estatico	LinStatic	Step By Step	1		Bottom	0	-42.7608	0
	Story1	Sismo Estatico	LinStatic	Step By Step	1		Top	0	-50.9302	0
	Story1	Sismo Estatico	LinStatic	Step By Step	1		Bottom	0	-50.9302	0

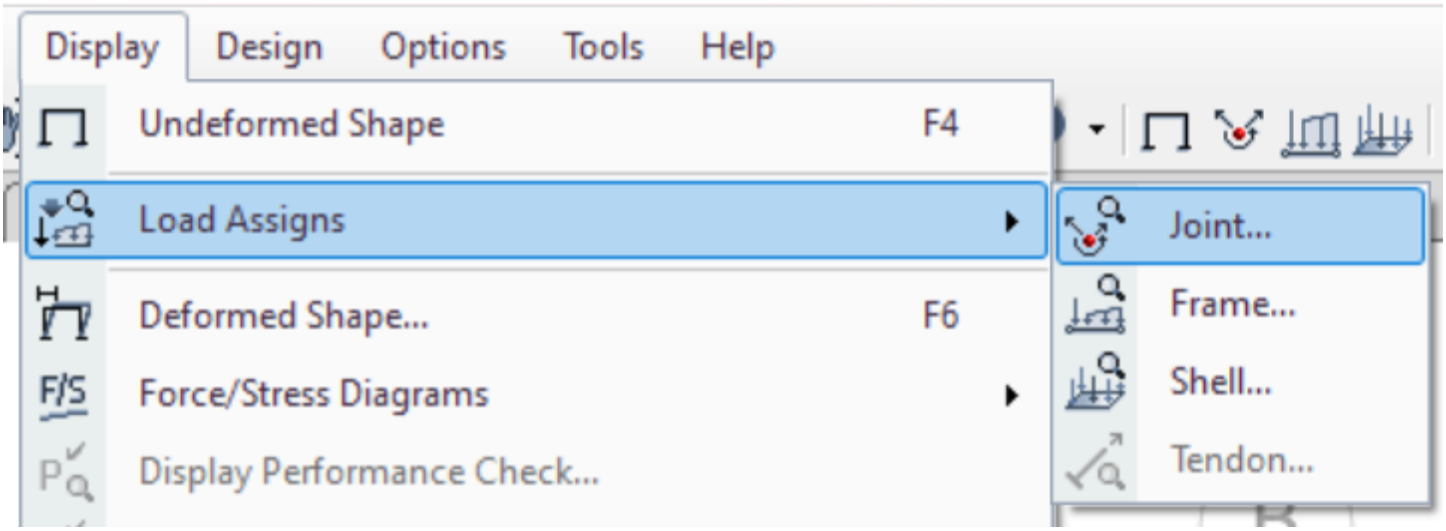
28.8. El story 3 queda con su mismo valor Vx
28.9. Story 2 = Vx - Vx 3
28.10. Story 1 = Vx1- Vx2

29. **PARA APLICARLE FUERZAS A CIERTOS PUNTOS.**



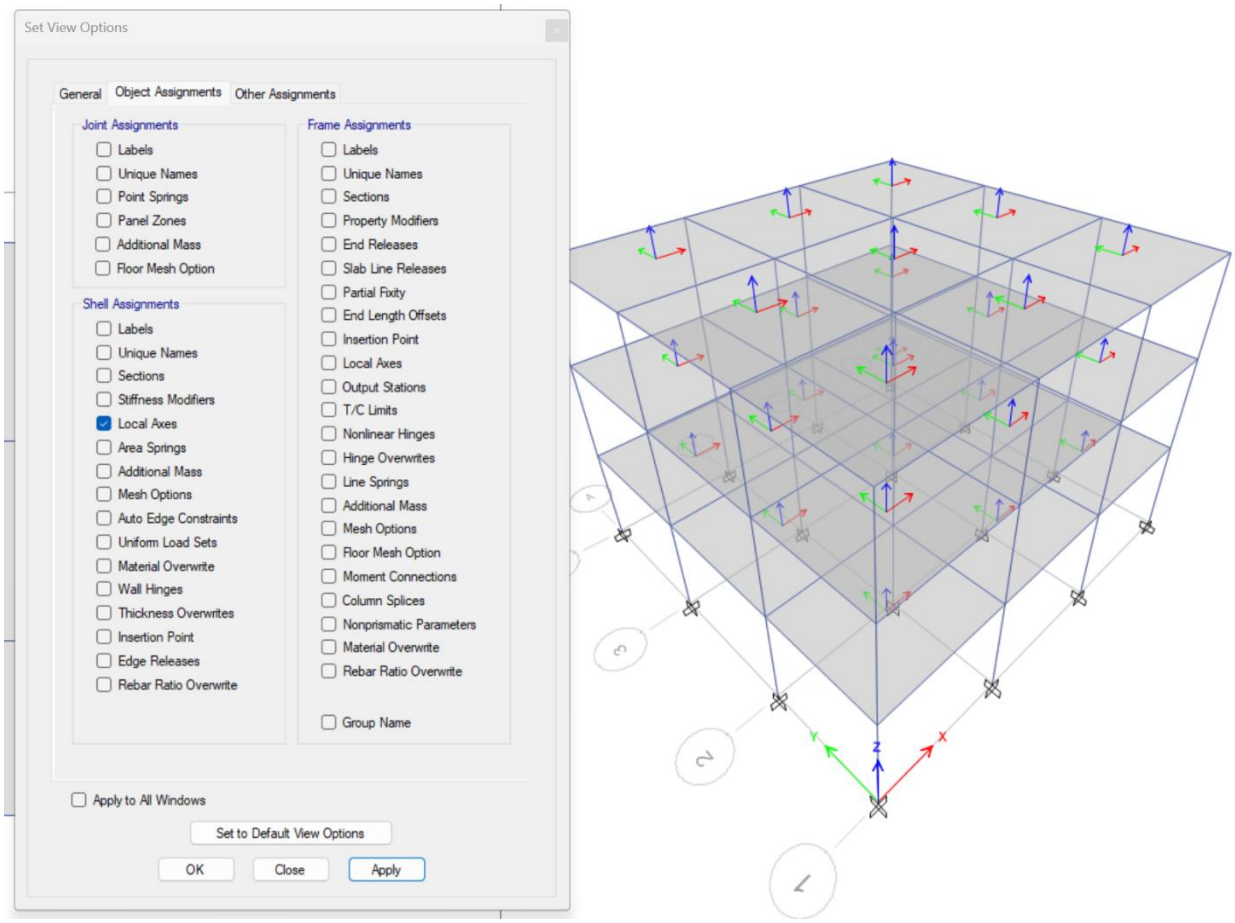
29.1. SELECT
29.2. Select
29.3. Labels

30. **DEZPLAZAMIENTOS.**



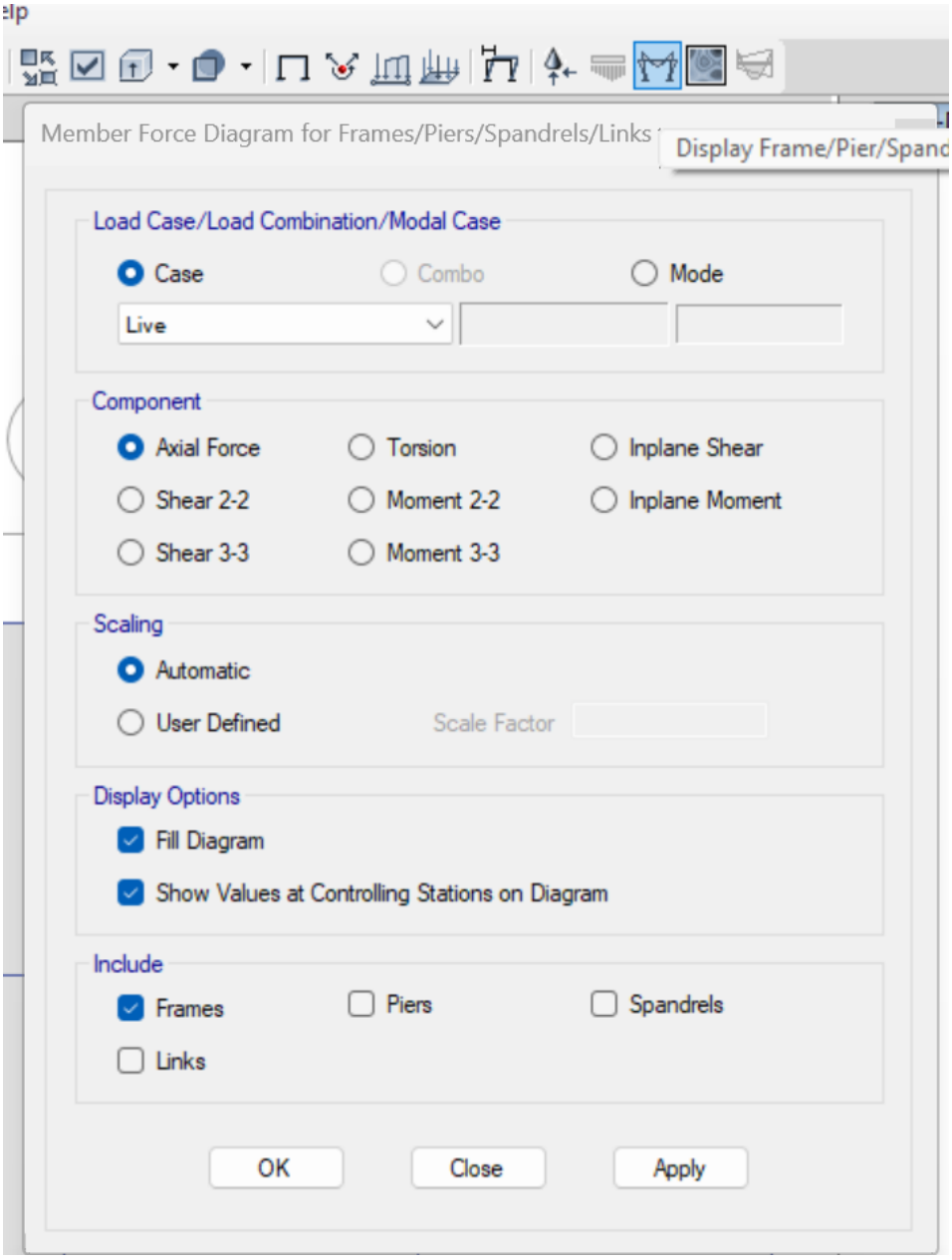
30.1. Sismo estático 1/6

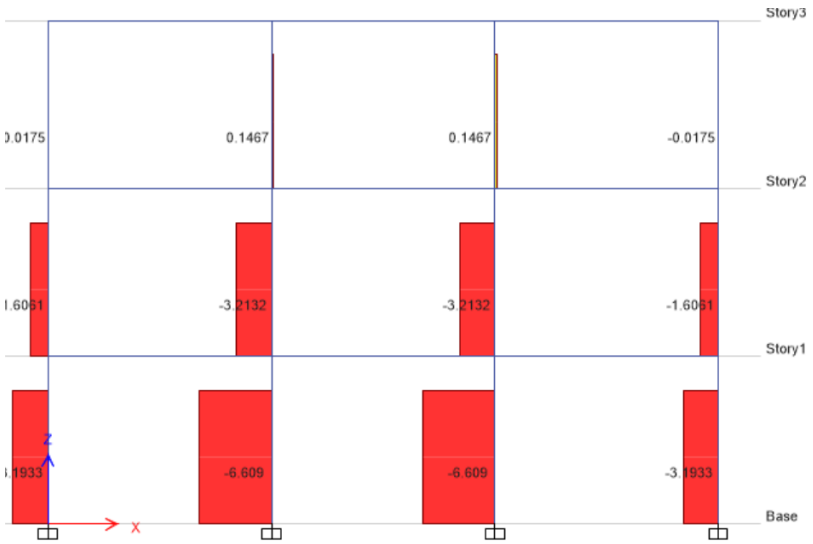
31. **AGREGAR EJES LOCALES**


Set View Options
General
Object Assignments
Other Assignments
Joint Assignments
Labels
Unique Names
Point Springs
Panel Zones
Additional Mass
Floor Mesh Option
Shell Assignments
Labels
Unique Names
Sections
Stiffness Modifiers
Local Axes
Area Springs
Additional Mass
Mesh Options
Auto Edge Constraints
Uniform Load Sets
Material Overwrite
Wall Hinges
Thickness Overwrites
Insertion Point
Edge Releases
Rebar Ratio Overwrite
Frame Assignments
Labels
Unique Names
Sections
Property Modifiers
End Releases
Slab Line Releases
Partial Fixity
End Length Offsets
Insertion Point
Local Axes
Output Stations
T/C Limits
Nonlinear Hinges
Hinge Overwrites
Line Springs
Additional Mass
Mesh Options
Floor Mesh Option
Moment Connections
Column Splices
Nonprismatic Parameters
Material Overwrite
Rebar Ratio Overwrite
Group Name
Apply to All Windows
Set to Default View Options
OK
Close
Apply
Y
Z
X

32. DIAGRAMA DE FUERZA EN FRAME AXIAL (EJE 1)

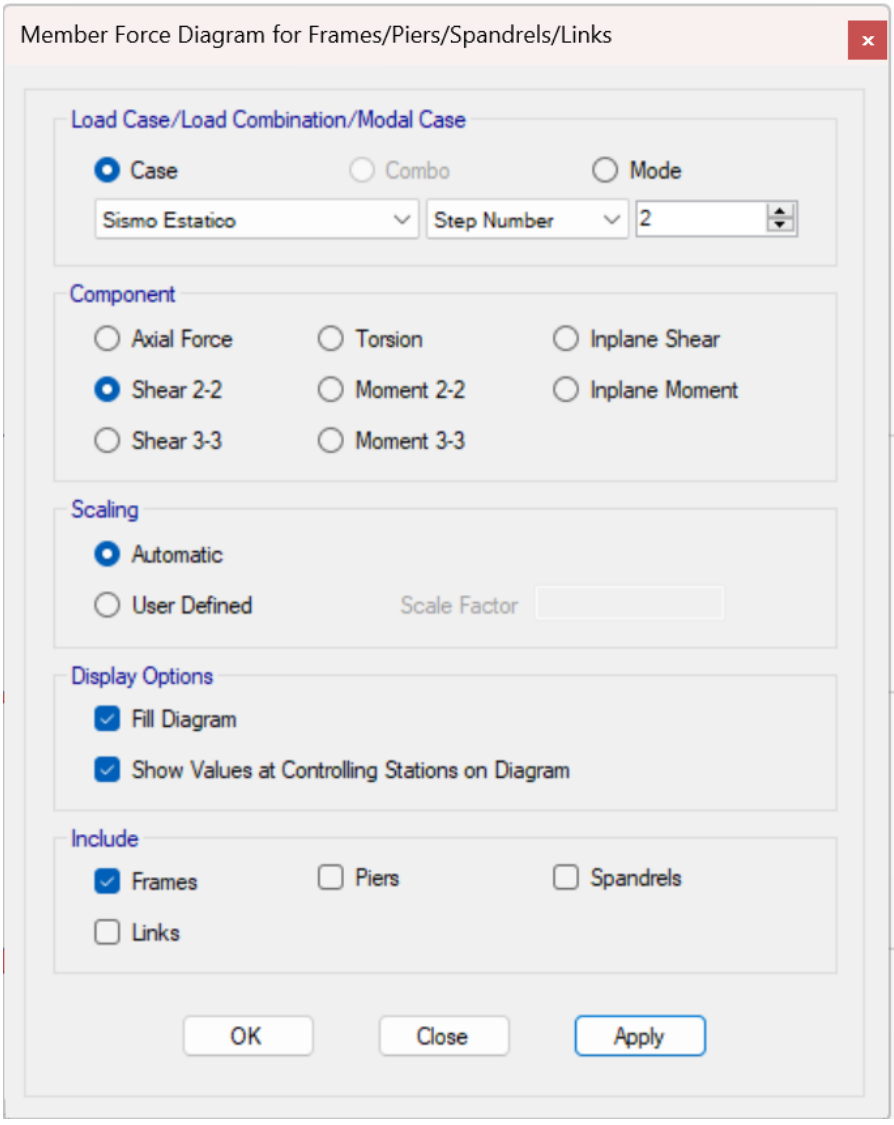
32.1. Debe estar en elevacion.

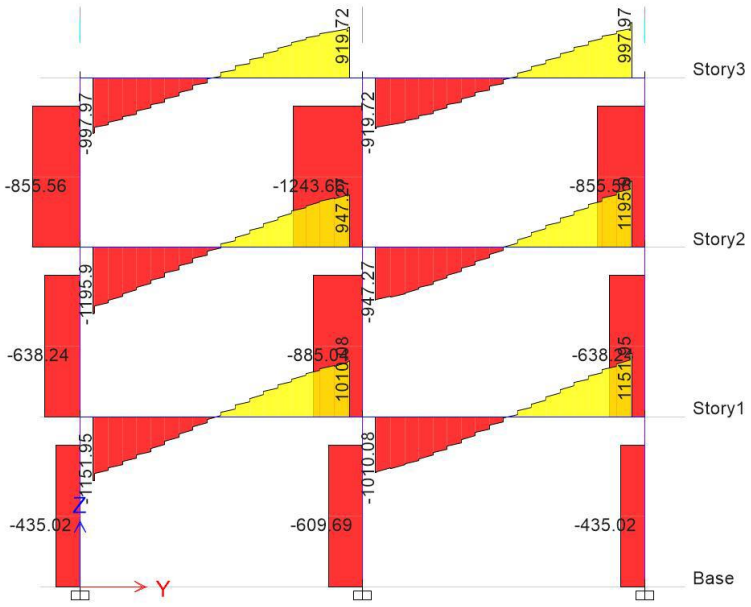




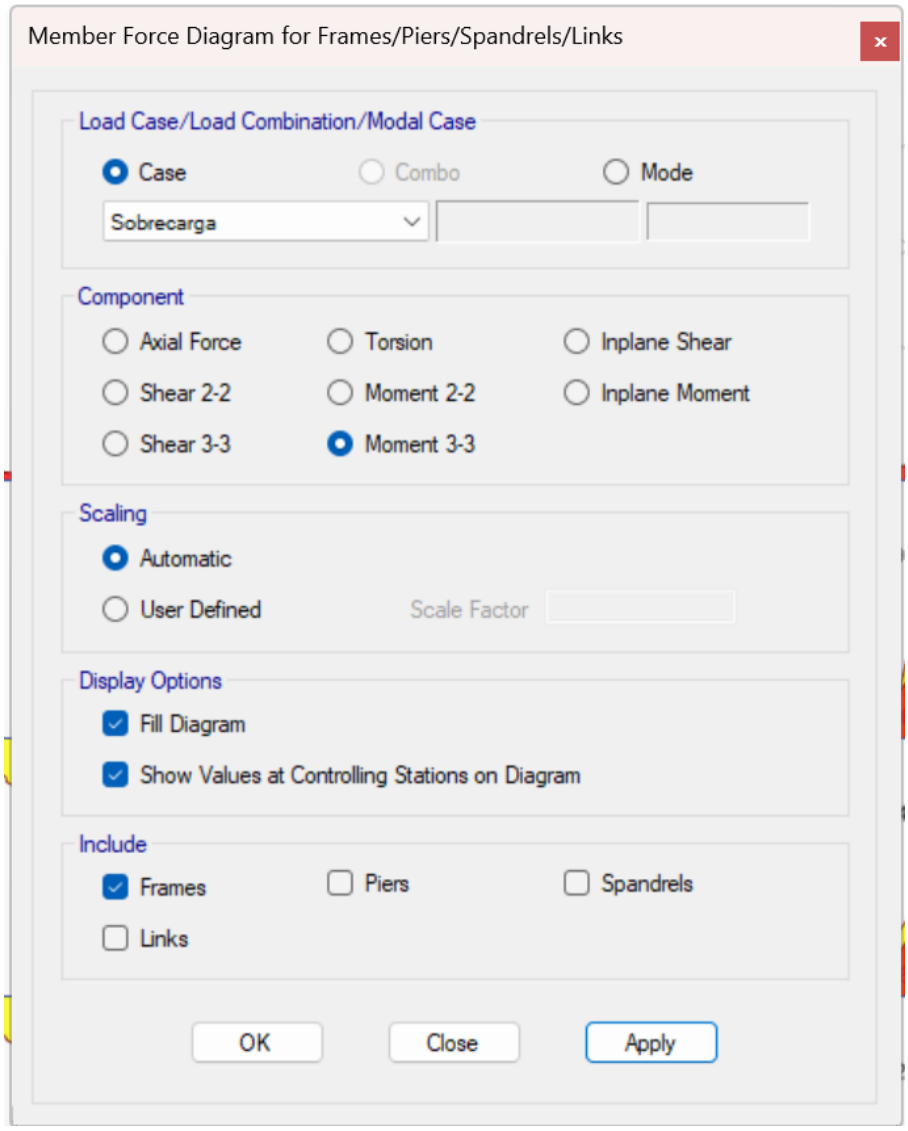
33. **DIAGRAMA DE CORTE EN FRAME (EJE 2 Y 3).**

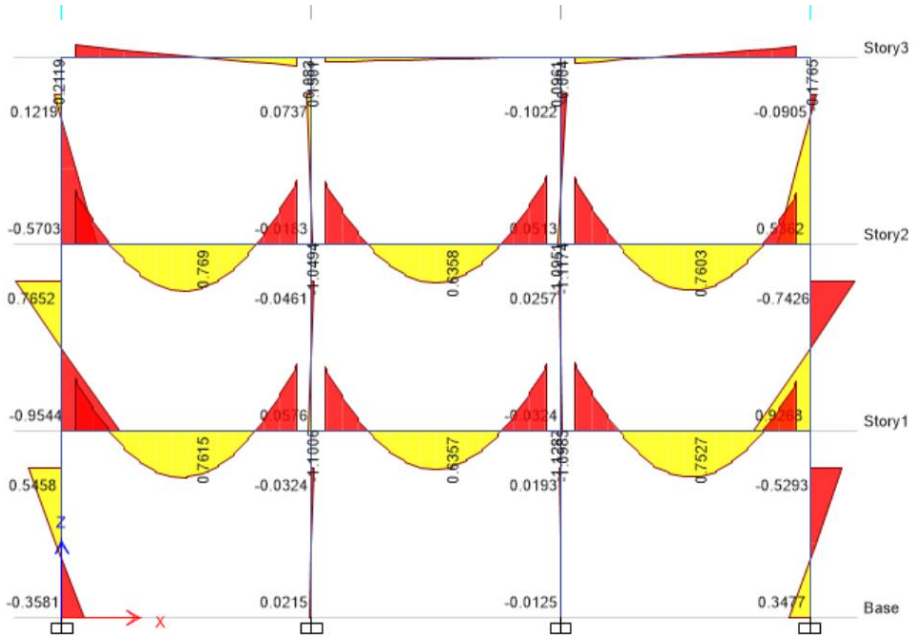
33.1. CASE: SISMOESTATICO.





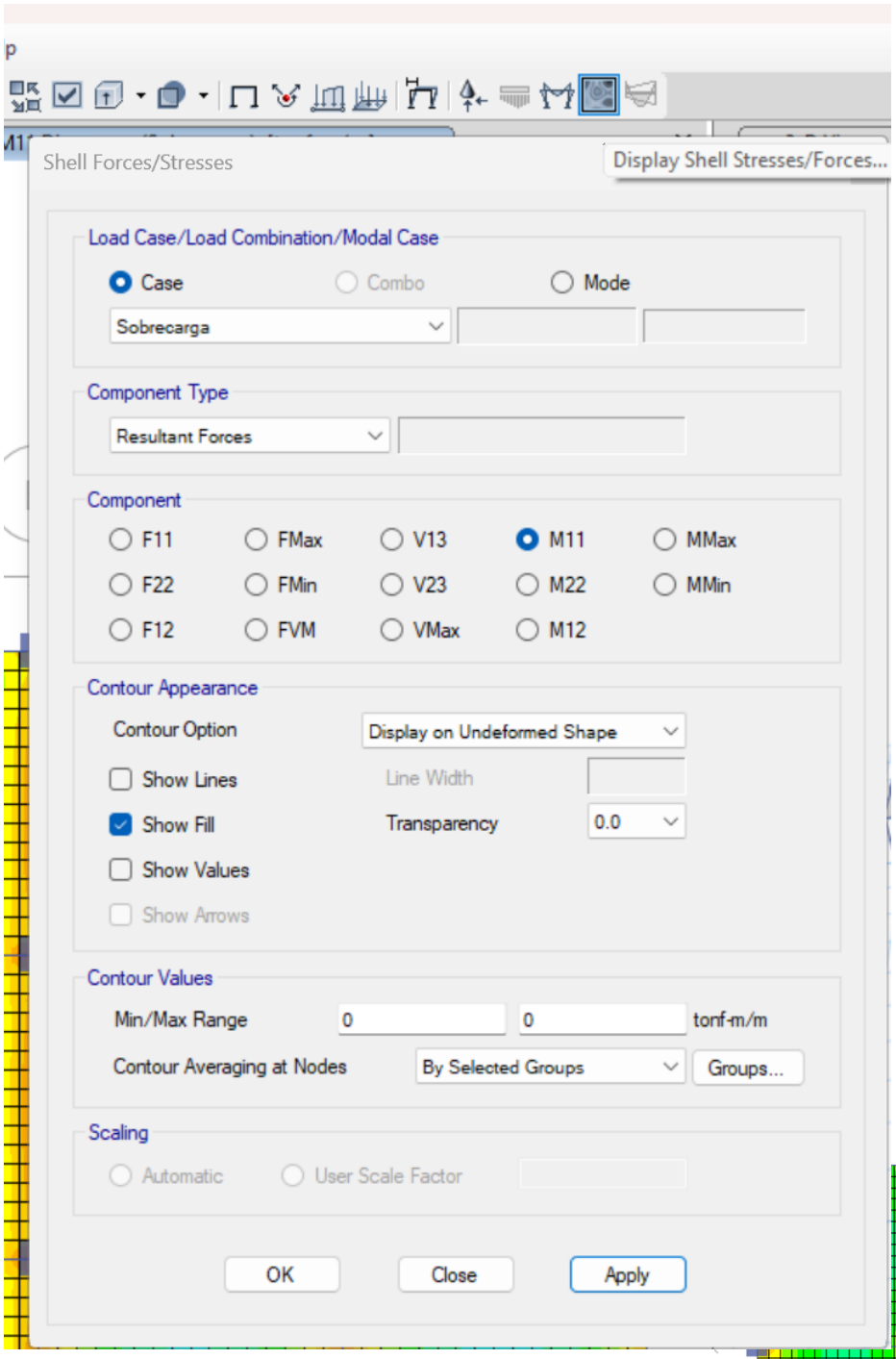
34. DIAGRAMA DE MOMENTO EN FRAME (EJE 2 Y 3)

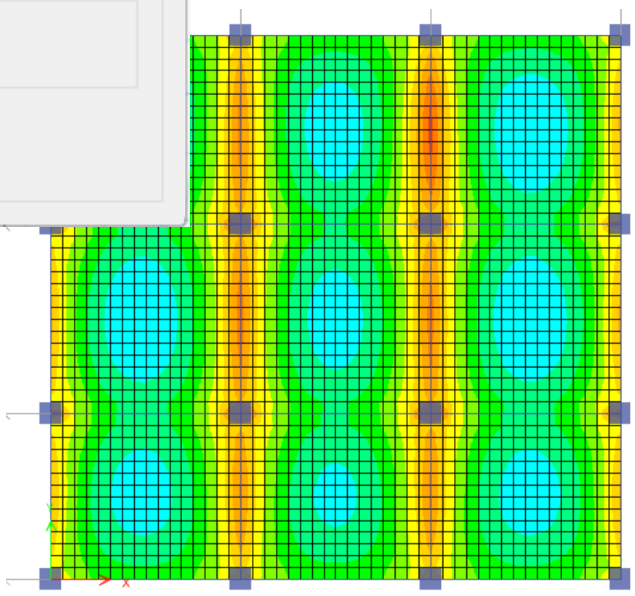


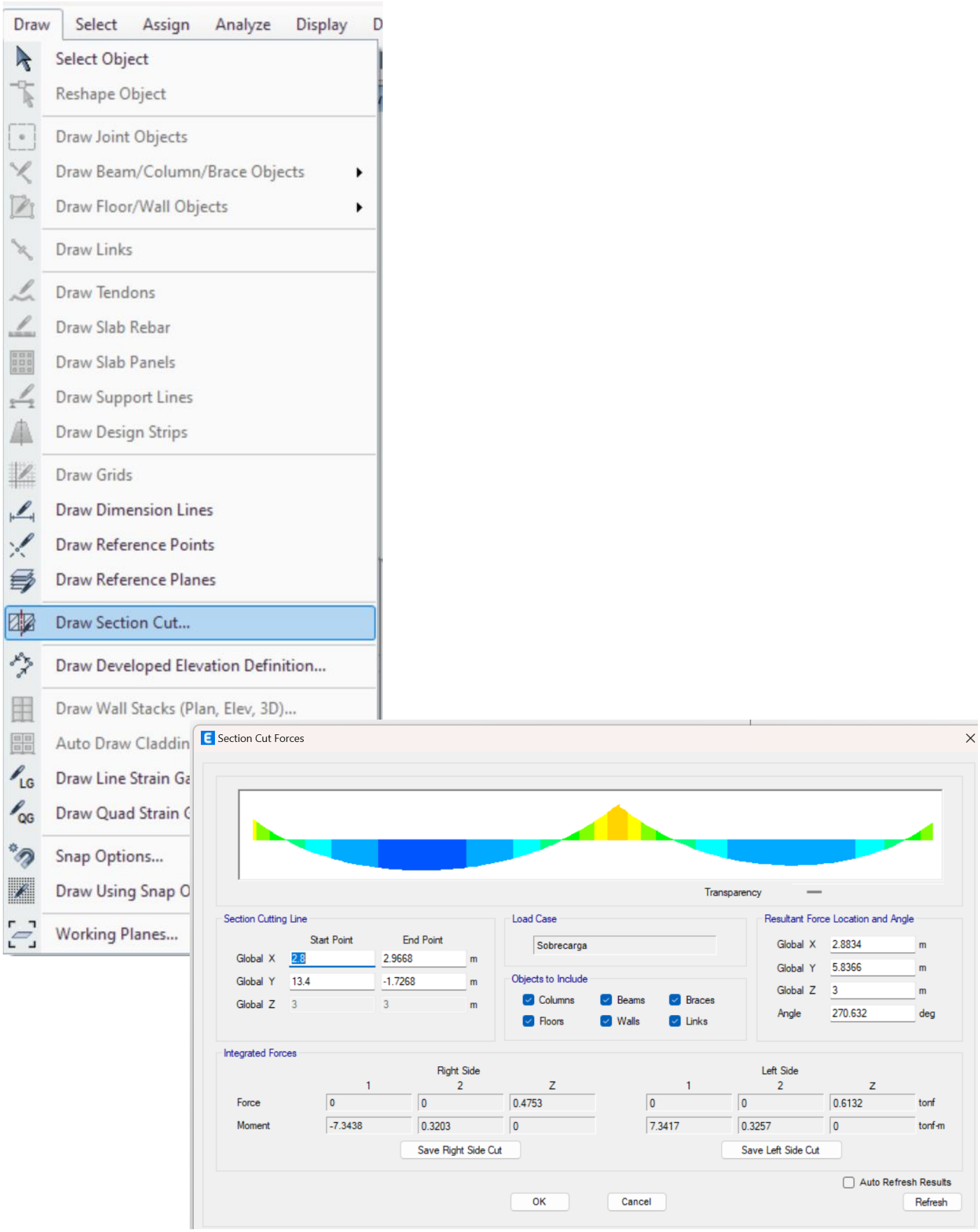


35. DIAGRAMA DE MOMENTO EN LOSA

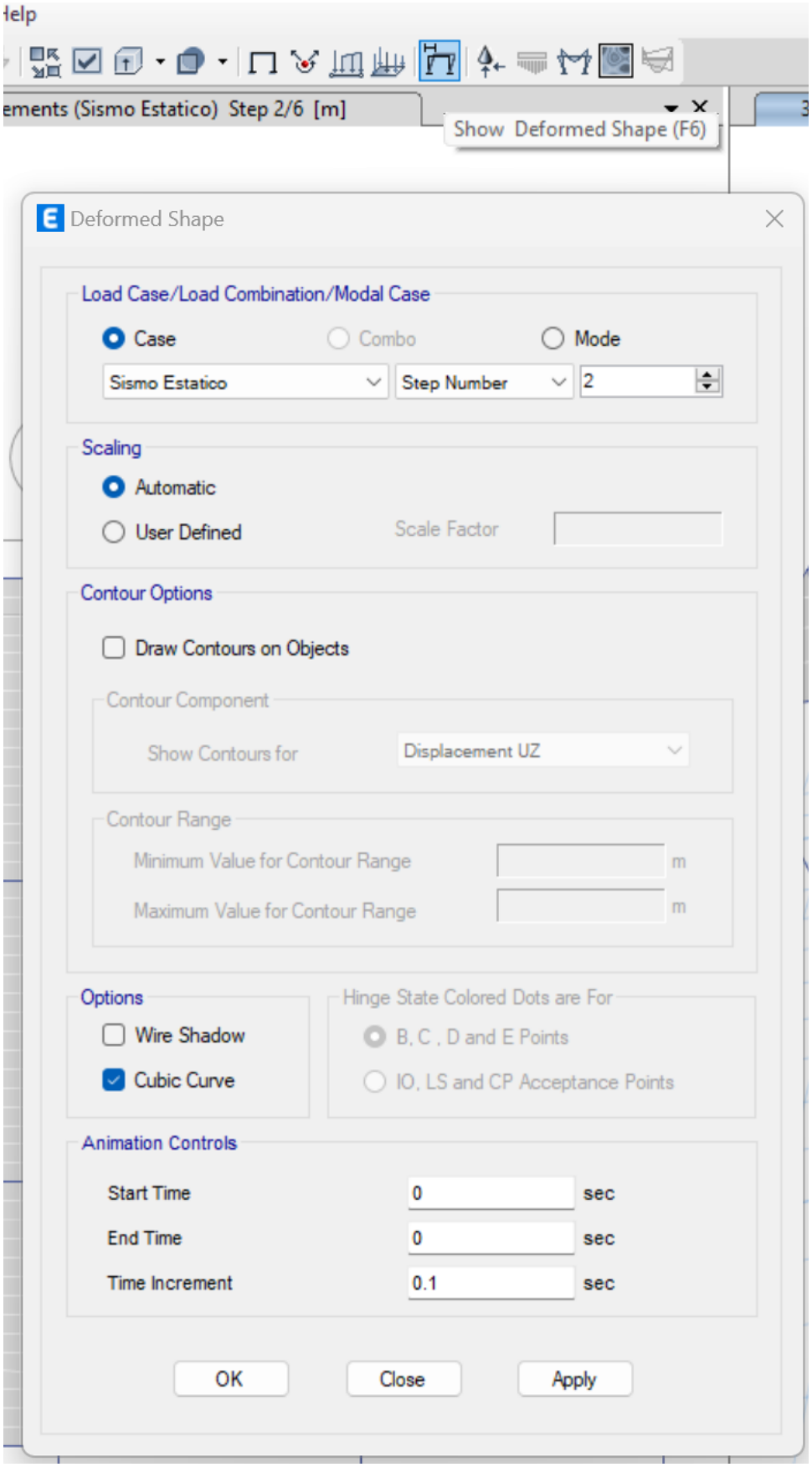
35.1. Debe estar en story





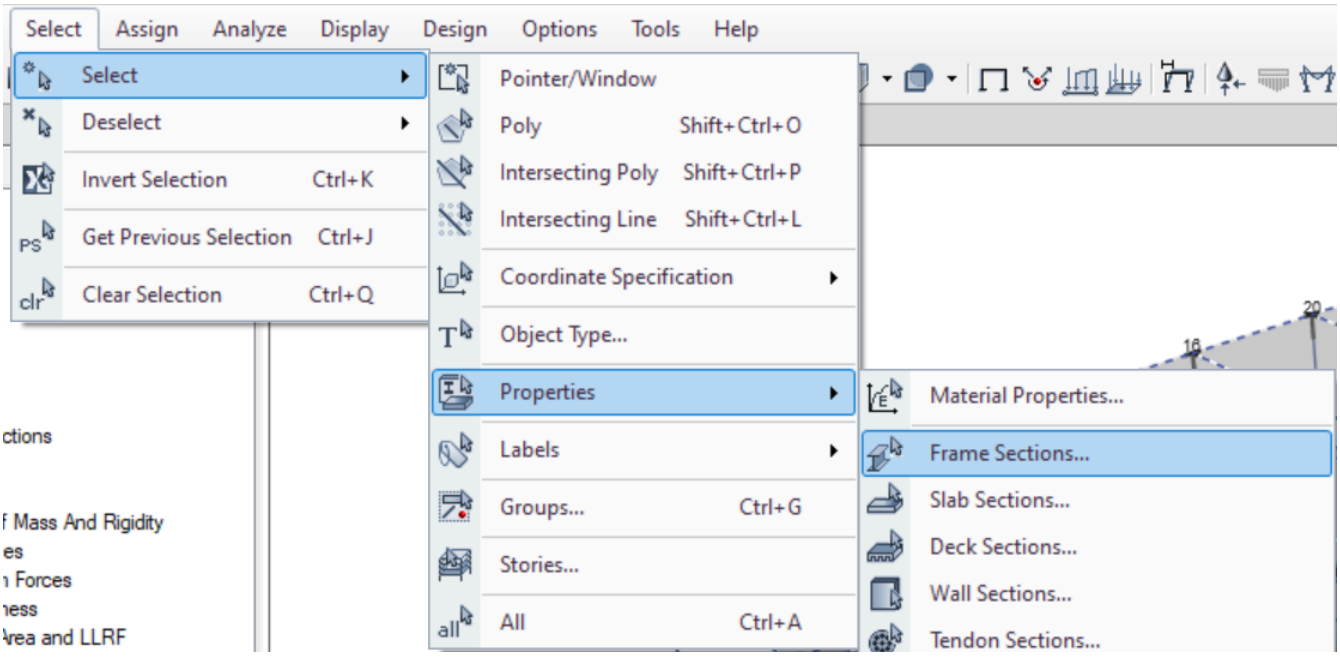

Draw
Select
Assign
Analyze
Display
Select Object
Reshape Object
Draw Joint Objects
Draw Beam/Column/Brace Objects
Draw Floor/Wall Objects
Draw Links
Draw Tendons
Draw Slab Rebar
Draw Slab Panels
Draw Support Lines
Draw Design Strips
Draw Grids
Draw Dimension Lines
Draw Reference Points
Draw Reference Planes
Draw Section Cut...
Draw Developed Elevation Definition...
Draw Wall Stacks (Plan, Elev, 3D)...
Snap Options...
Working Planes...
Section Cut Forces
Transparency
Section Cutting Line
Start Point
End Point
Global X
2.8
2.9668
m
Global Y
13.4
-1.7268
m
Global Z
3
3
m
Load Case
Sobrecarga
Objects to Include
Columns
Beams
Braces
Floors
Walls
Links
Resultant Force Location and Angle
Global X
2.8834
m
Global Y
5.8366
m
Global Z
3
m
Angle
270.632
deg
Integrated Forces
Right Side
Left Side
1
2
Z
Force
0
0
0.4753
0
0
0.6132
tonf
Moment
-7.3438
0.3203
0
7.3417
0.3257
0
tonf-m
Save Right Side Cut
Save Left Side Cut
Auto Refresh Results
OK
Cancel
Refresh

36. DESPLAZAMIENTO DE NODOS



37. **AGRIETAMIENTO DE VIGAS Y COLUMNAS.**

37.1. SELECT
37.2. Select
37.3. Properties
37.4. Frame Section.
37.5. Seleccionar las vigas y columnas por aparte.



37.6. ASIGN.
37.7. Frame
37.8. Properties Modifiers.

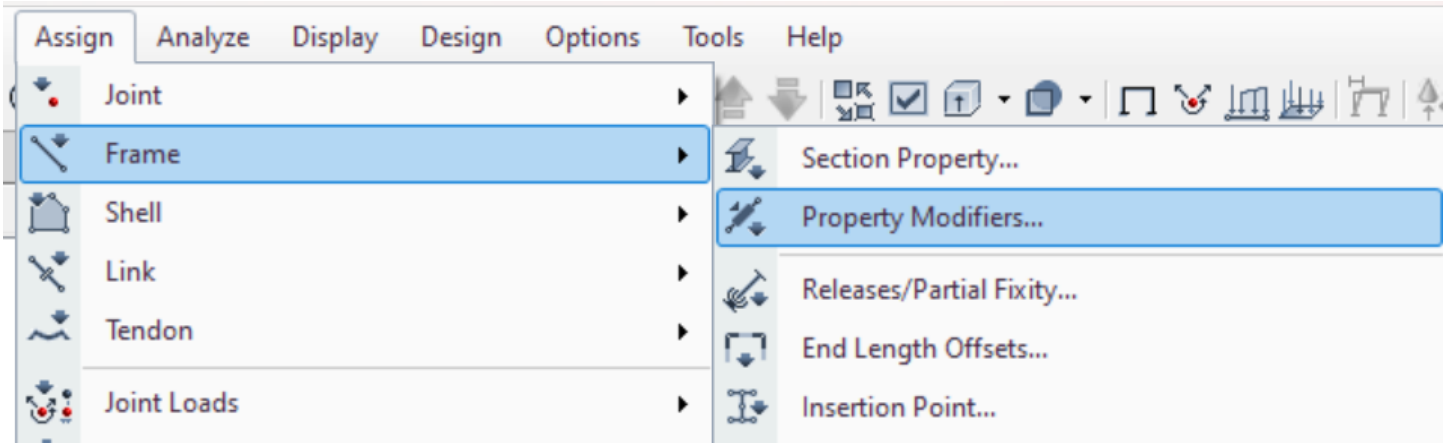
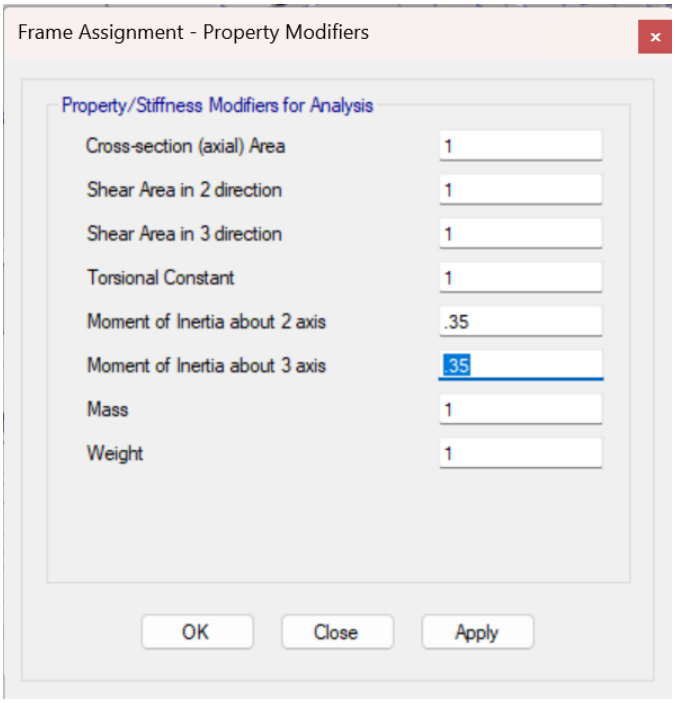


Table 6.6.3.1.1(a)—Moments of inertia and cross-sectional areas permitted for elastic analysis at factored load level

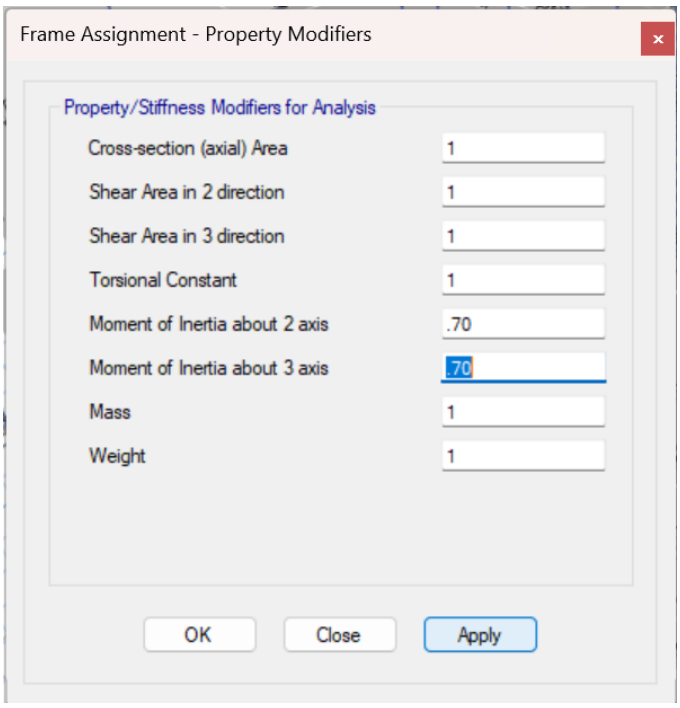
Member and condition		Moment of inertia	Cross-sectional area for axial deformations	Cross-sectional area for shear deformations
Columns		$0.70I_g$	$1.0A_g$	$b_w h$
Walls	Uncracked	$0.70I_g$		
	Cracked	$0.35I_g$		
Beams		$0.35I_g$		
Flat plates and flat slabs		$0.25I_g$		

37.9. **Vigas=0.35**
37.10. **Columnas=0.70**

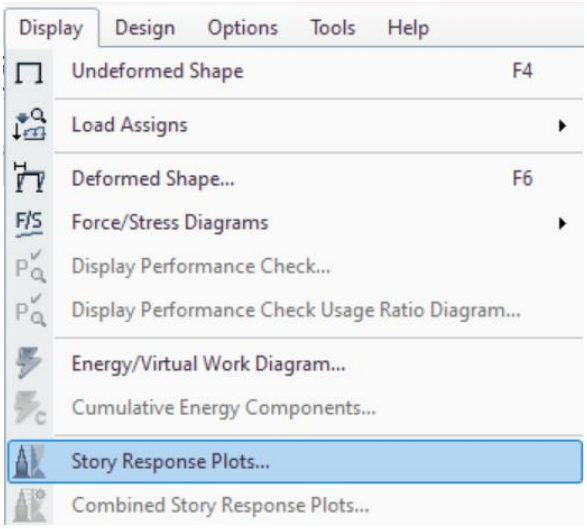
VIGAS



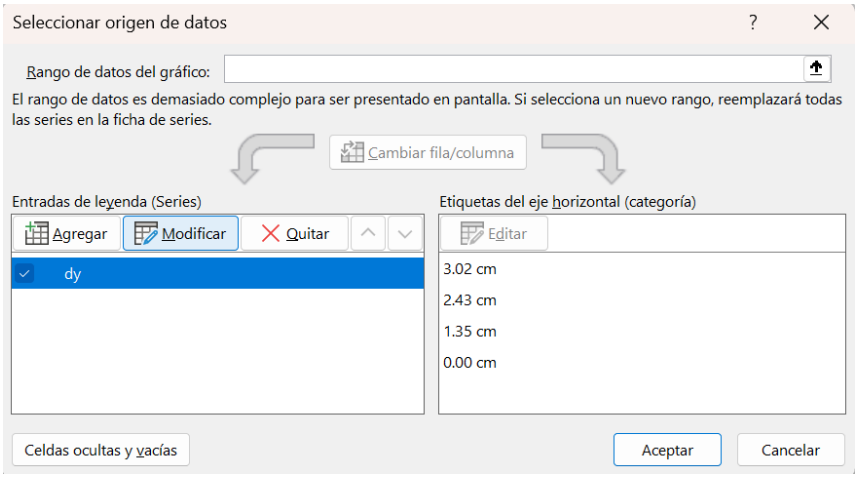
COLUMNAS

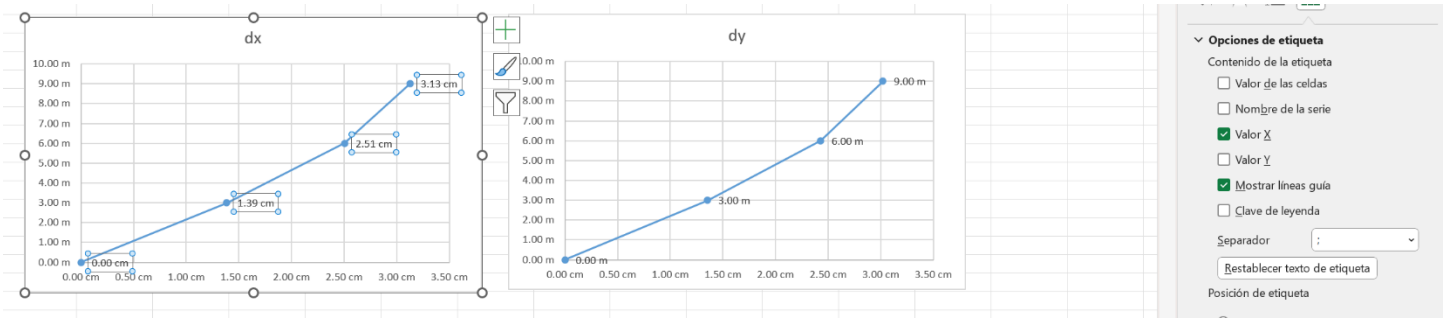


38. **PARA DESPLAZAMIENTOS LATERALES.**
    38.1. Ubicarnos en STORYY 1
    38.2.
39. **PARA DERIVAS.**
    39.1. Para verificar los valores

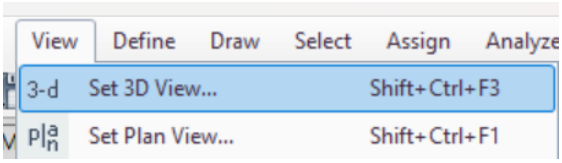


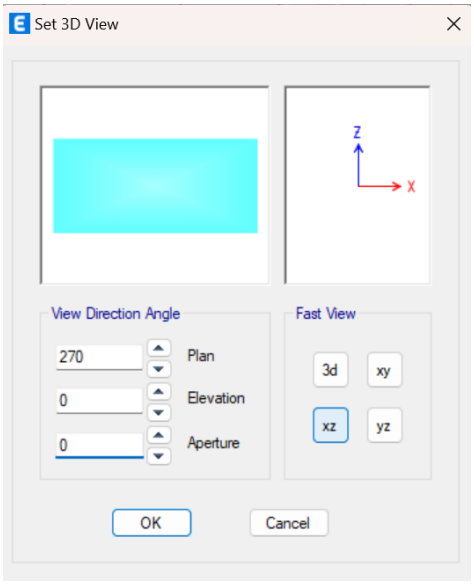
    39.2. Por cada Story se busca el Ux y el Uy de valor mayor y lo marcamos de rojo

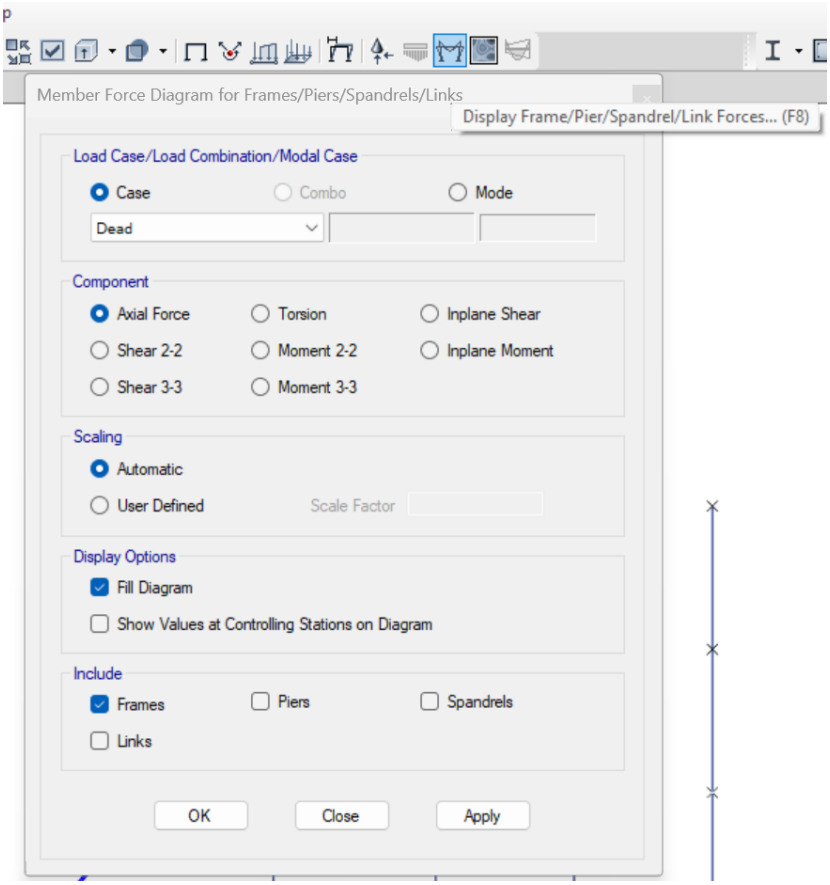


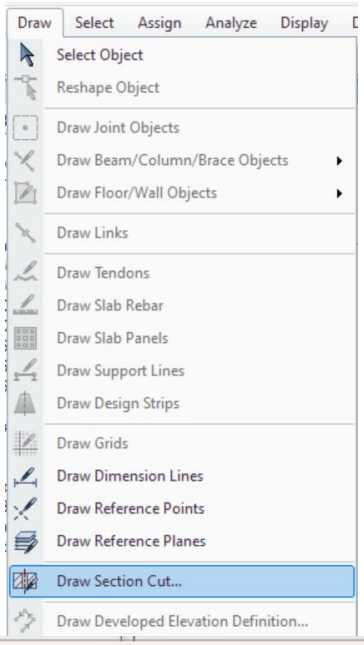


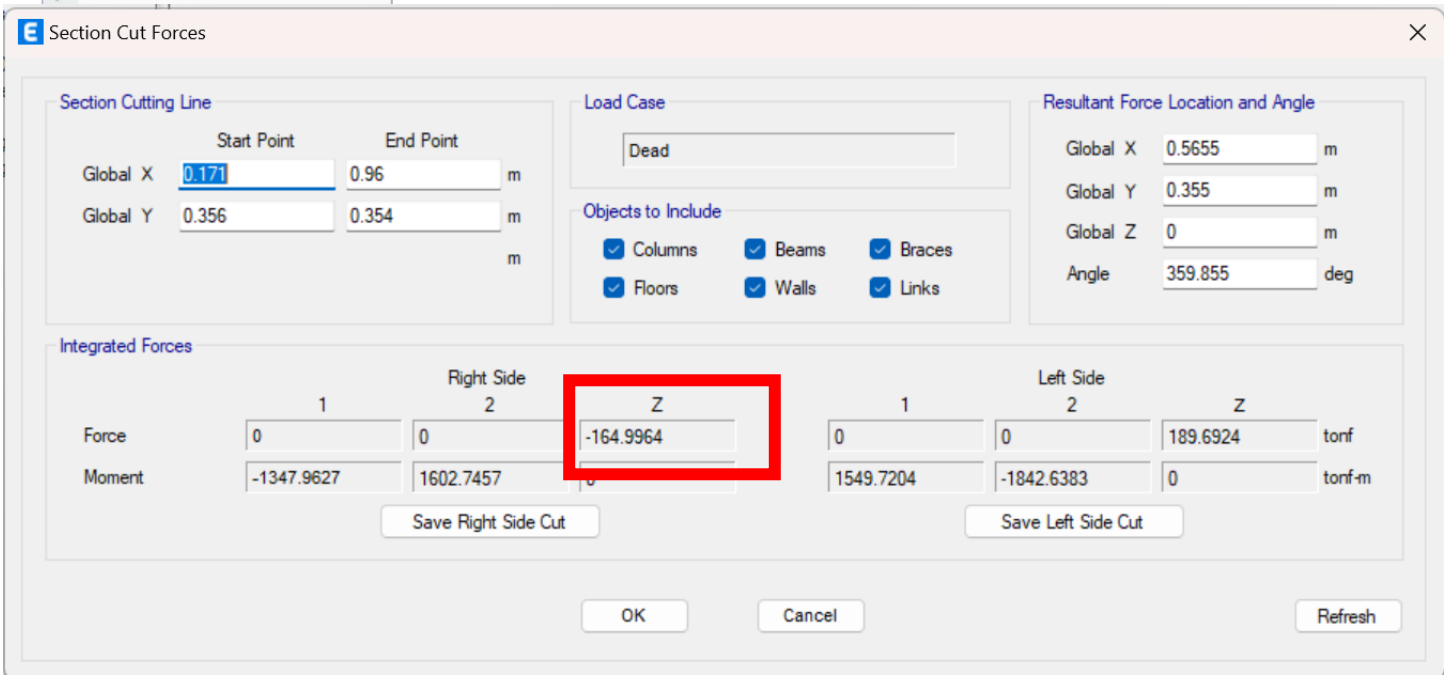
40. **PARA EL MOMENTO ACUMULADO.**
    - 40.1. VIEW
    - 40.2. Set 3Dview.
    - 40.3. Xz
    - 40.4. Angulo de apertura = 0





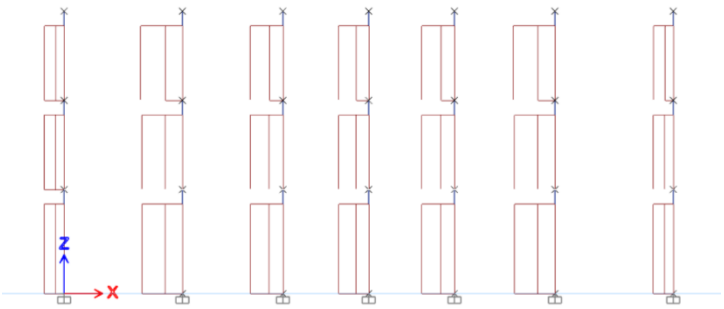

Member Force Diagram for Frames/Piers/Spandrels/Links
Display Frame/Pier/Spandrel/Link Forces... (F8)
Load Case/Load Combination/Modal Case
Case
Combo
Mode
Dead
Component
Axial Force
Torsion
Inplane Shear
Shear 2-2
Moment 2-2
Inplane Moment
Shear 3-3
Moment 3-3
Scaling
Automatic
User Defined
Scale Factor
Display Options
Fill Diagram
Show Values at Controlling Stations on Diagram
Include
Frames
Piers
Spandrels
Links
OK
Close
Apply





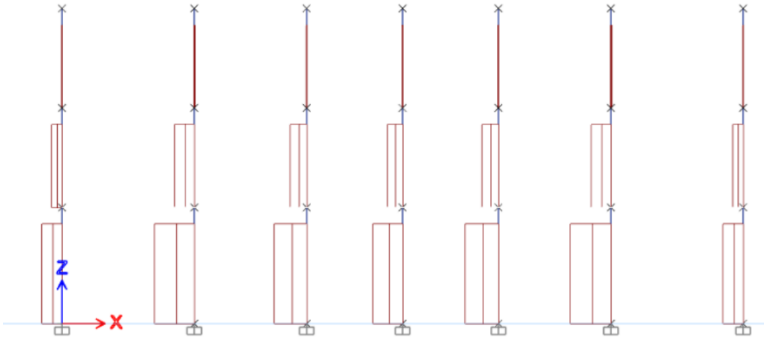
41. **VIVA DE TECHO ACUMULADA.**

41.1. Si se elije la viva de techo será la misma para todos los niveles porque solo esta en el techo.



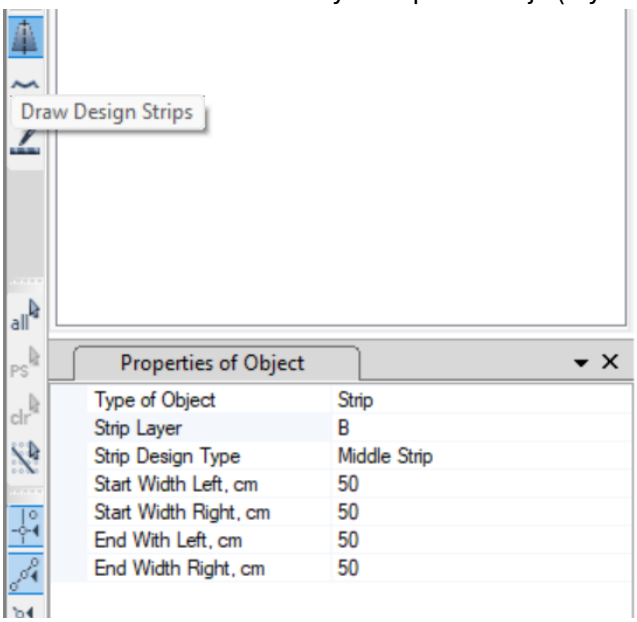
NIVEL	Vtacumuladas	Vt
3	70.04 Ton	70.04 Ton
2		
1		

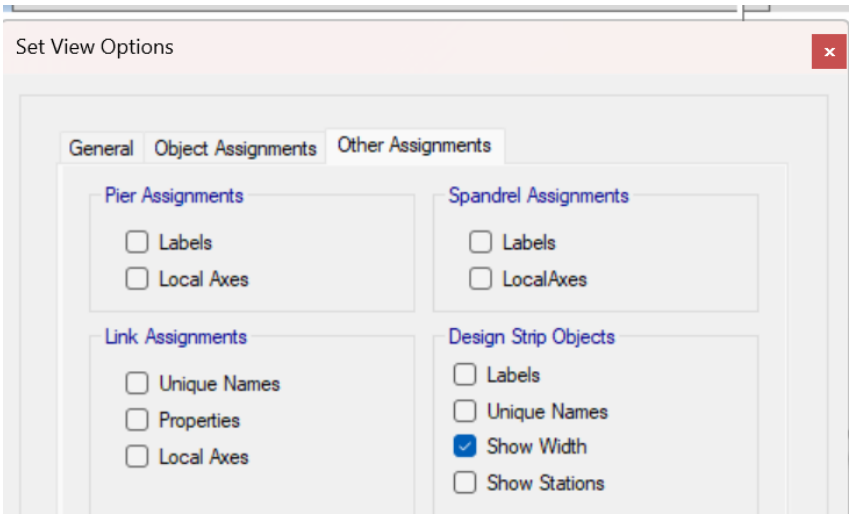
42. **VIVA ACUMULADA.**



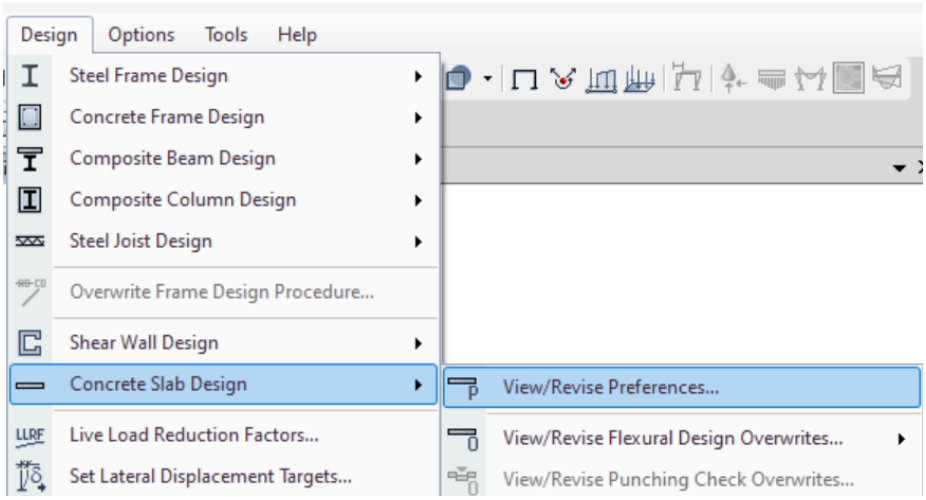
43. **DISEÑO DE LOSA.**

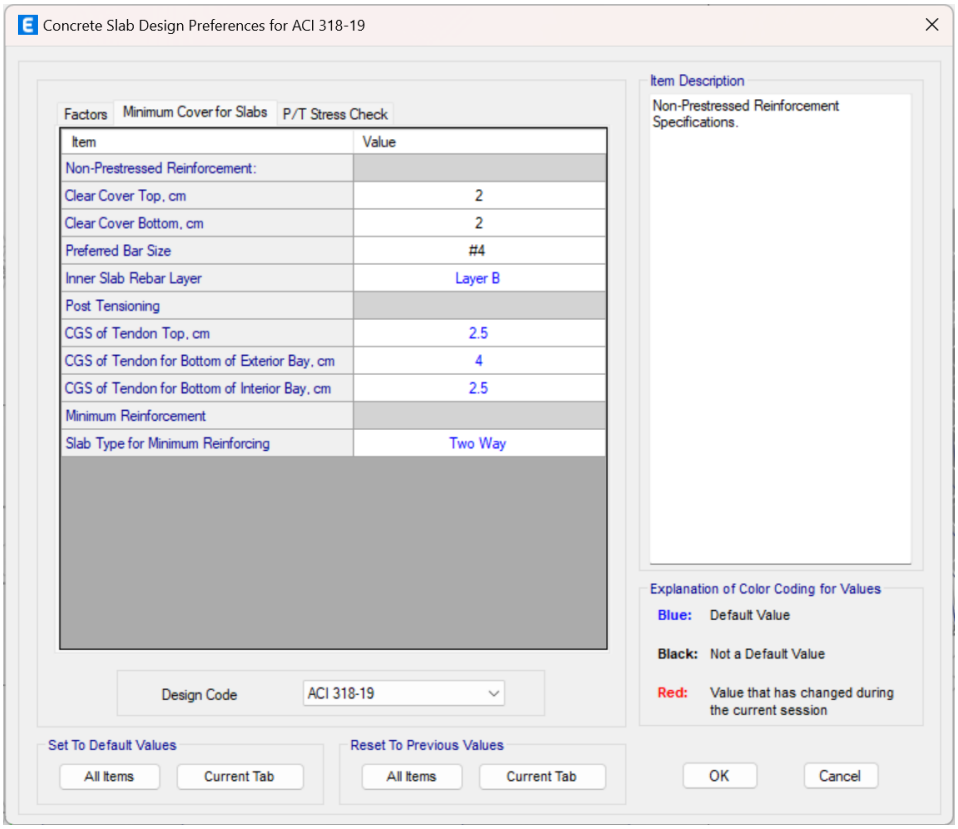
43.1. Cada Layer es para un eje (layer A = eje x)



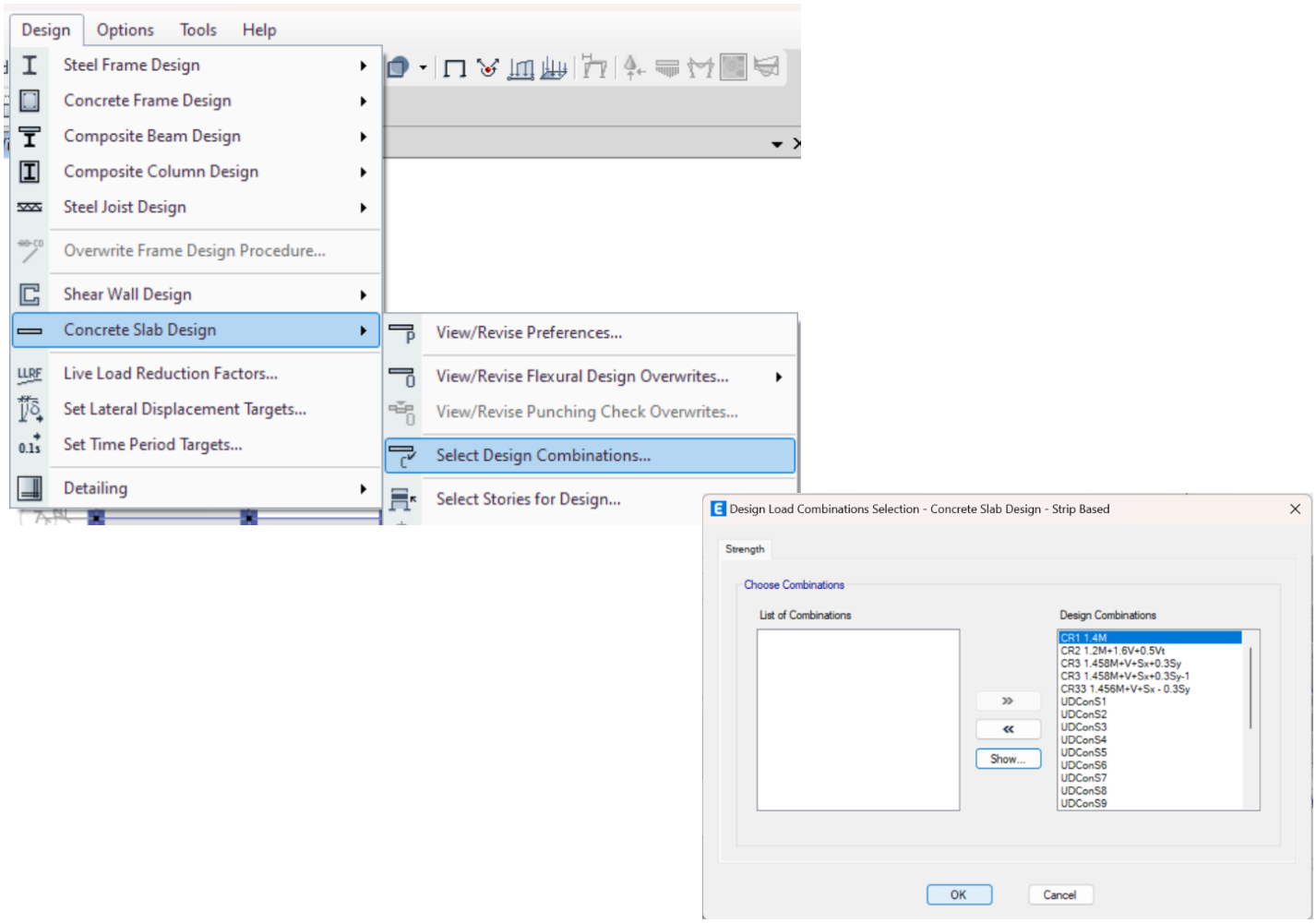


43.2. Revisar el acero y recubrimiento.

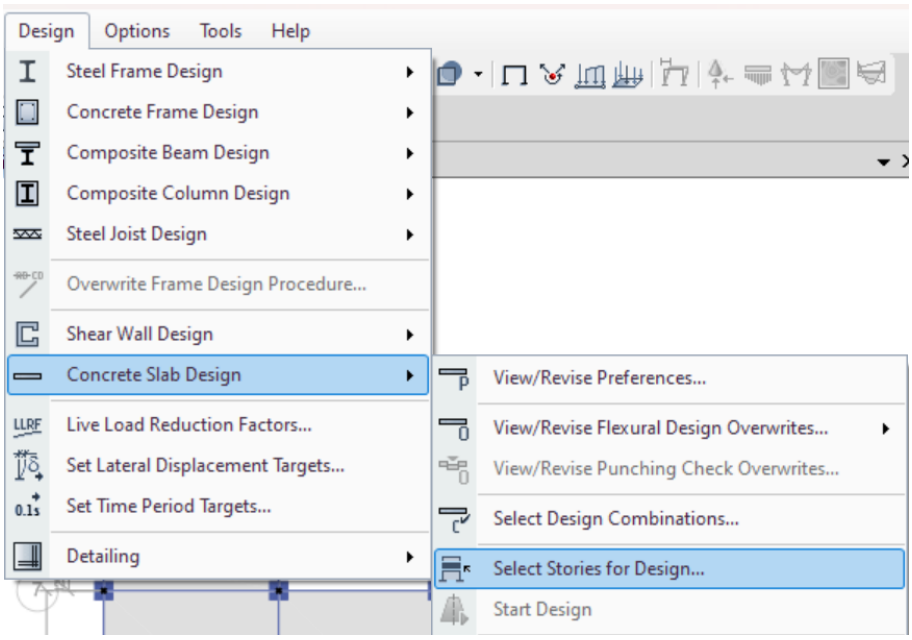


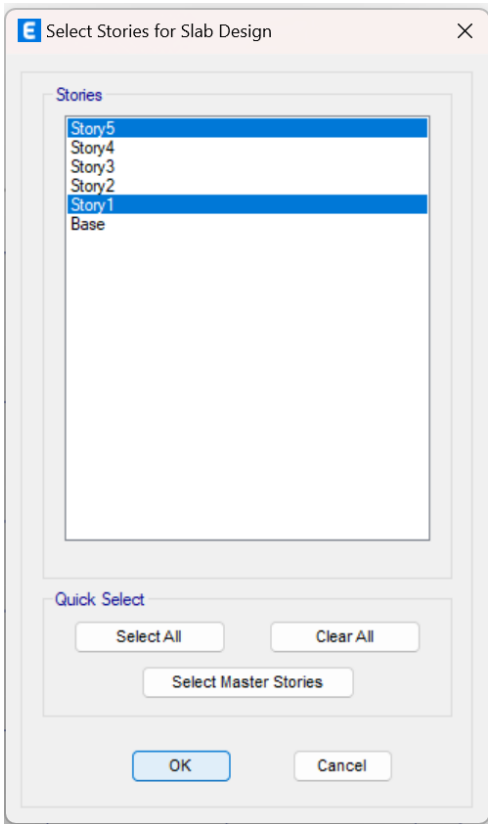


43.3. Ver los combos de diseño.



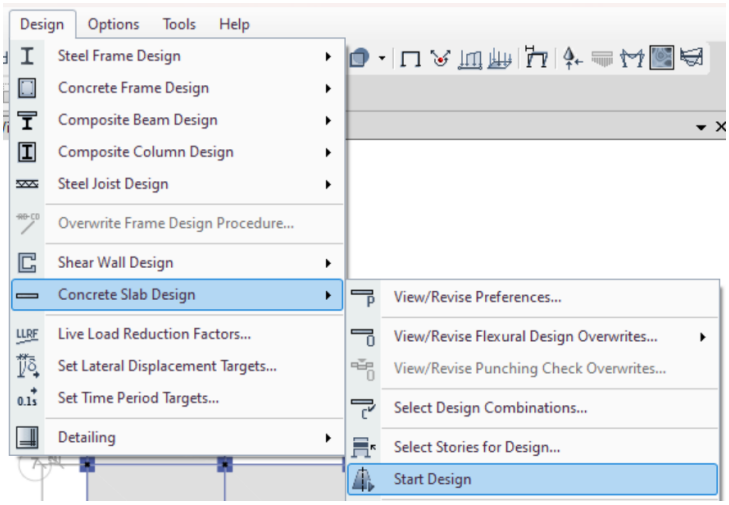
43.4. Elegir el nivel que se desea analizar.



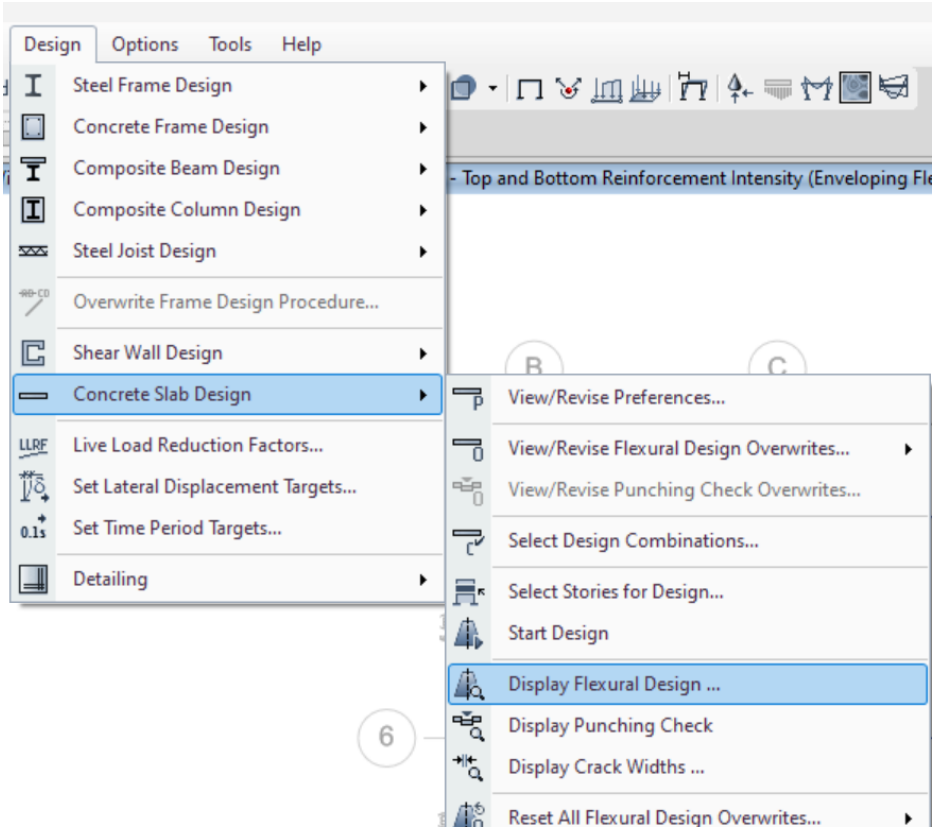


43.5. **Correr el programa.**

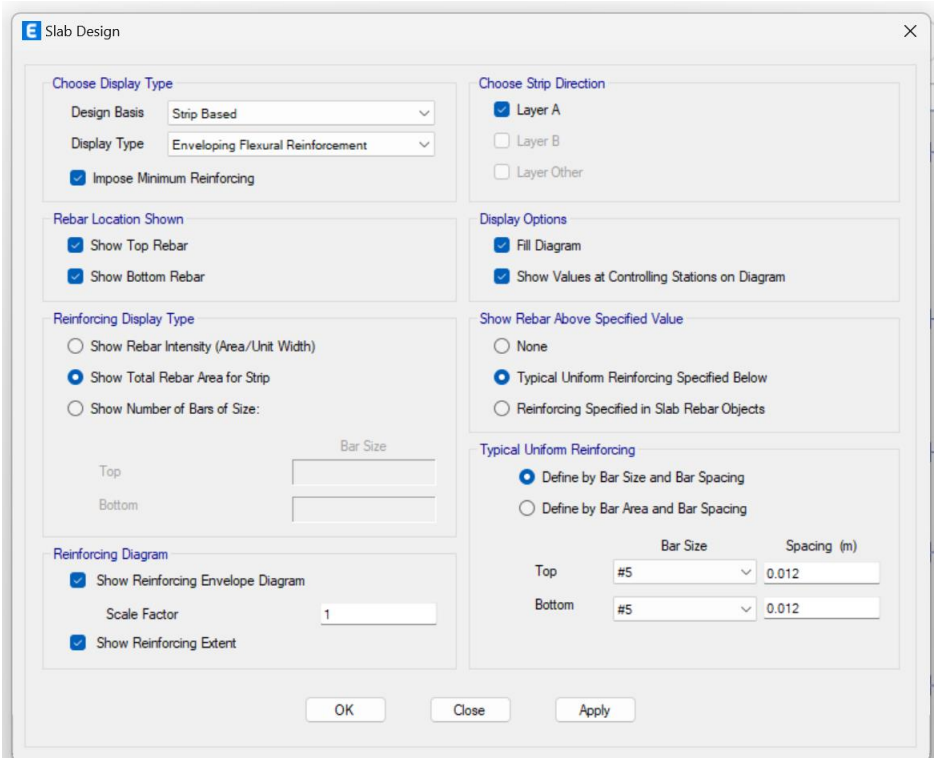
43.6. **Empezar diseño.**



43.7. Para chequear el acero.



43.8. CHEQUEMOS LA DISTANCIA Y NUMERO DE VARILLA QUE DESAPAREZCA EL DIAGRAMA DE LA LOSA.



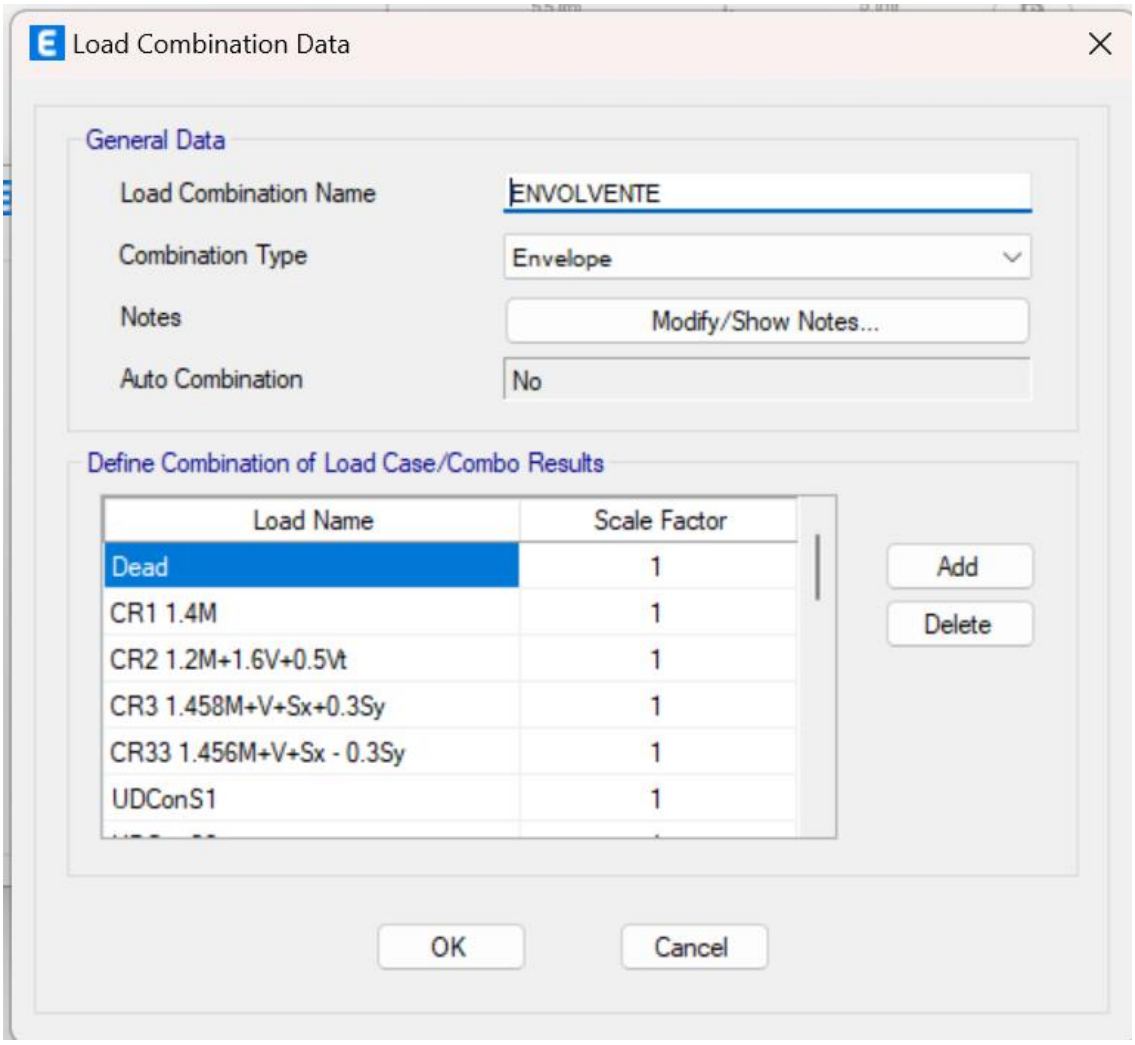
44. **DISEÑO DE VIGA.**

44.1. **DEFINE**

44.2. Combinación de carga (Load Combinations)

44.3. Add New Combo > tipo envolvente > las 4 gravitacionales y los 20 creados automáticamente =1

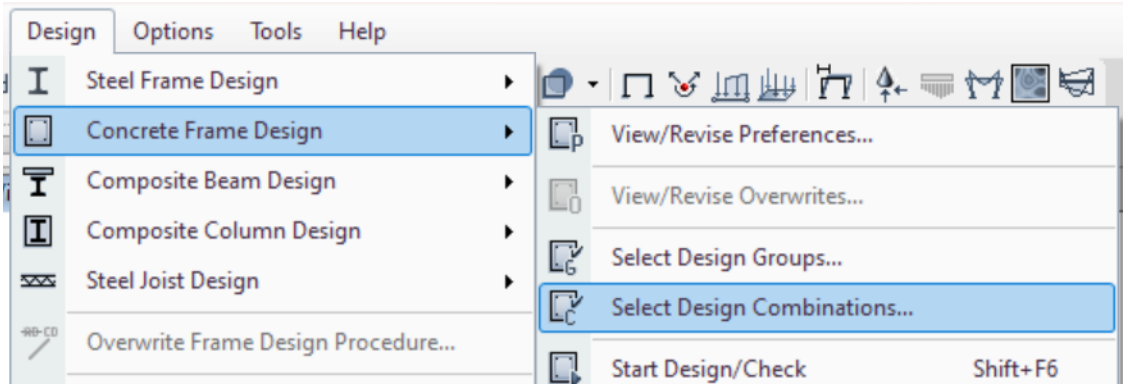
44.4. **CORRER EL PROGRAMA**



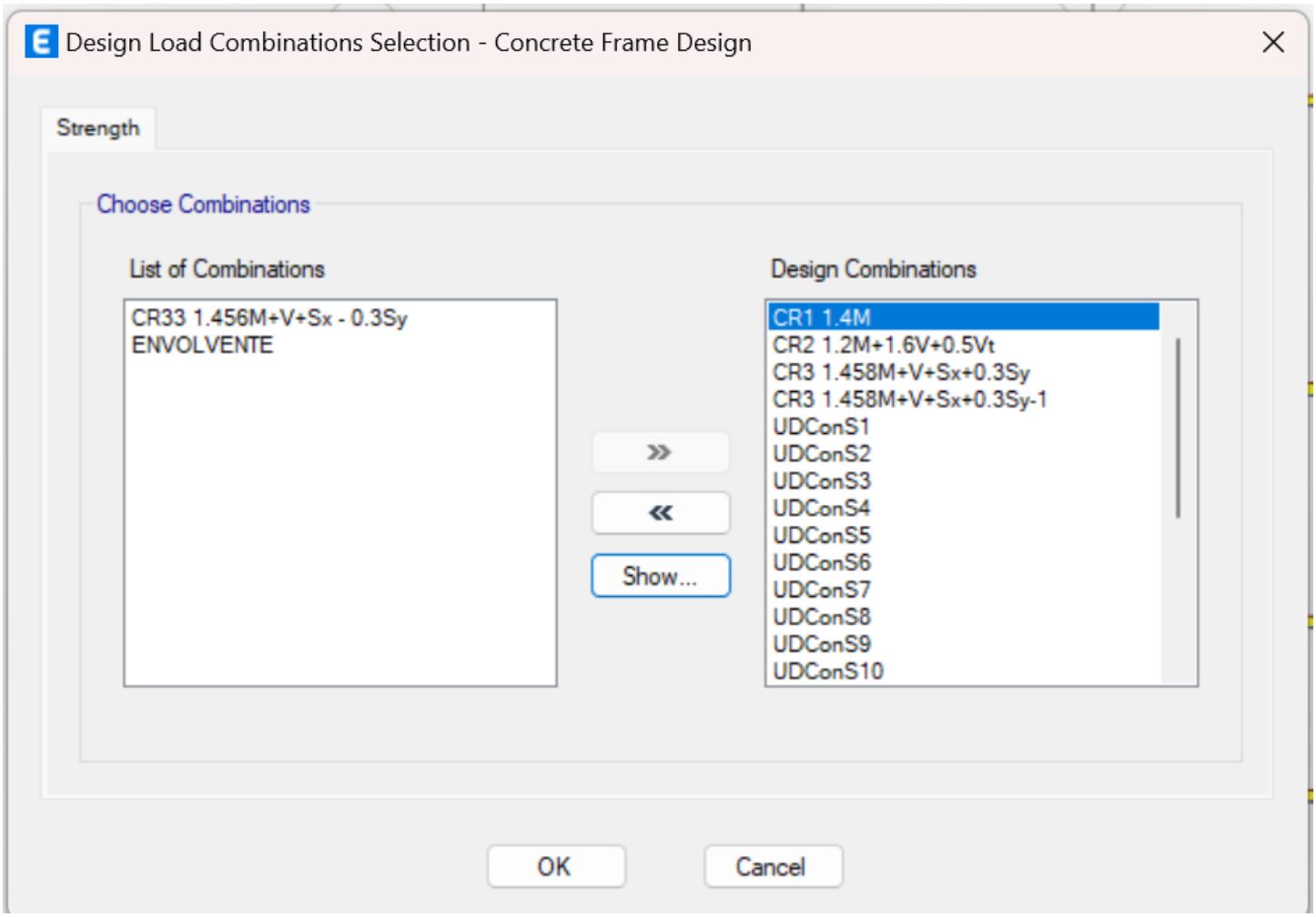
44.5. Presentar el diagrama de Momentos, pero respecto a la envolvente.

44.6. eso para ver los valores min y max

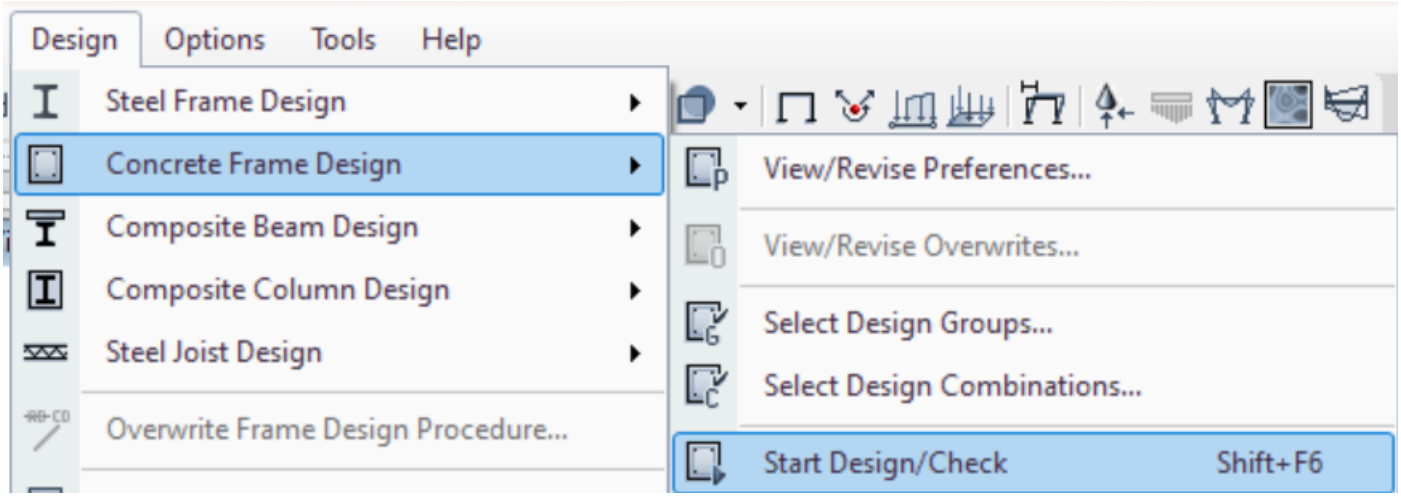
44.7. **DESIGN**



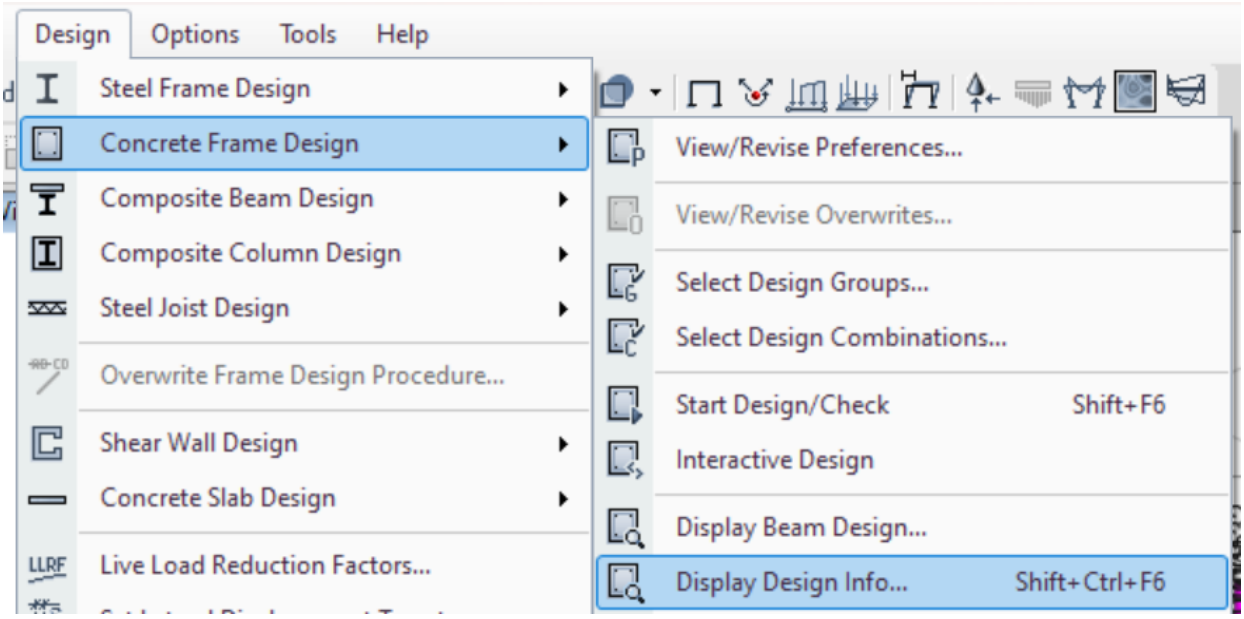
AGREGO LAS QUE HAGAN FALTA MENOS LA ENVOLVENTE PORQUE YA ESTAN TODAS LAS COMBINACIONES DEL OTRO LADO

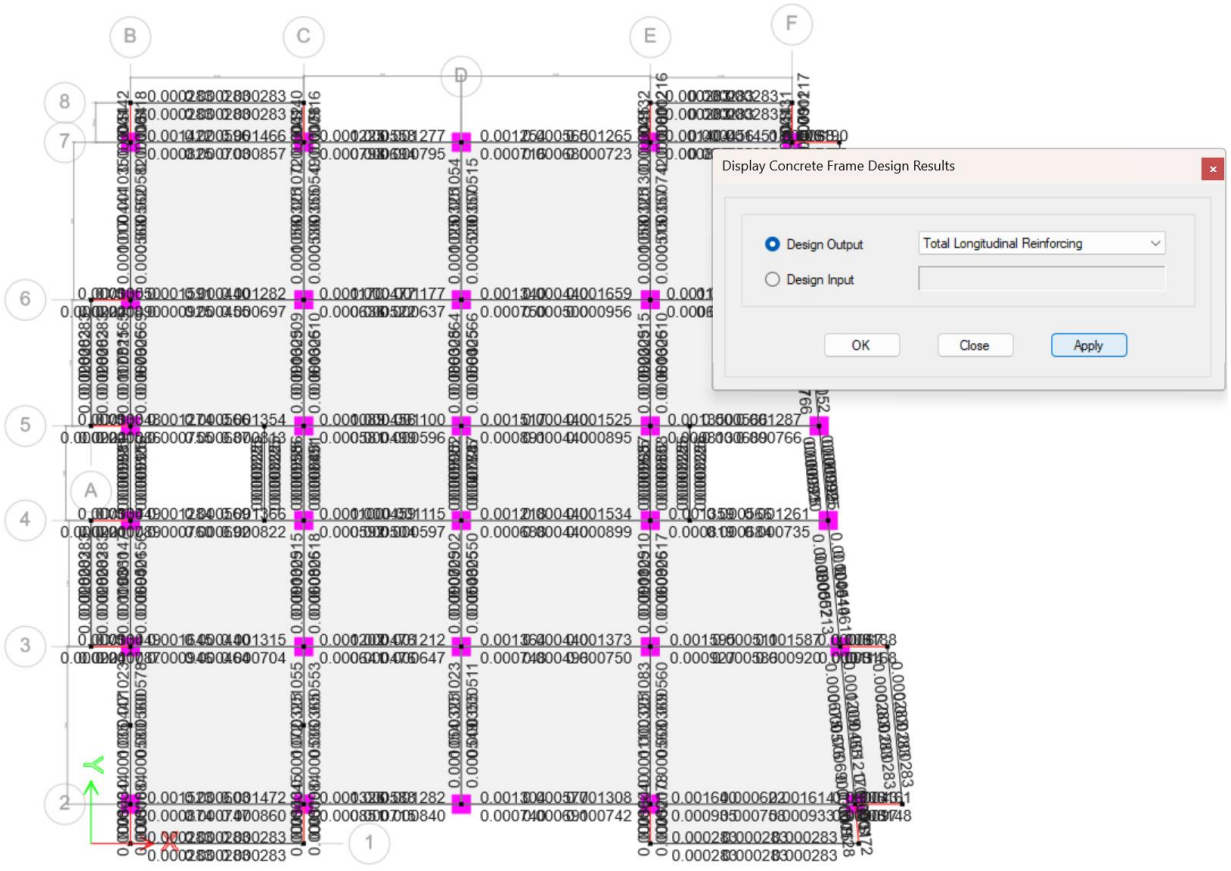


44.8. INICIAMOS EL DISEÑO DE LA VIGA.



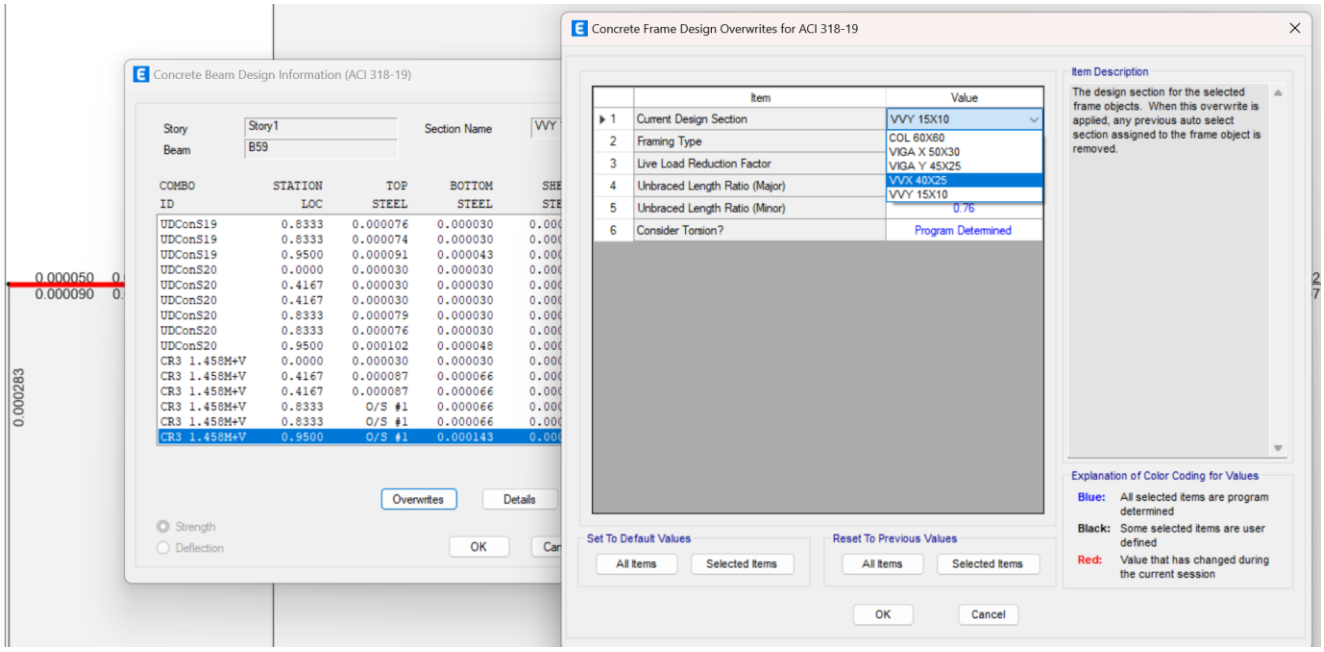
44.9. Ya chequeados las vigas se procede a revisar la información



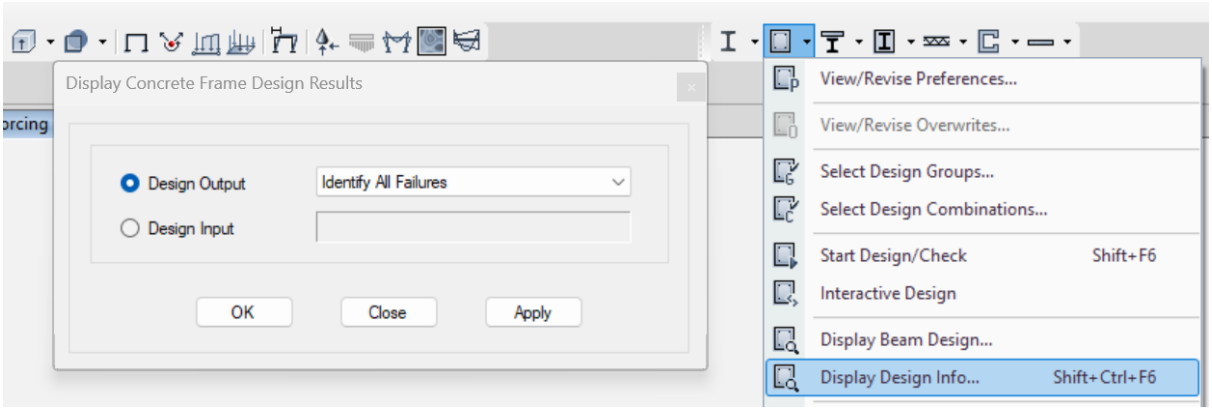


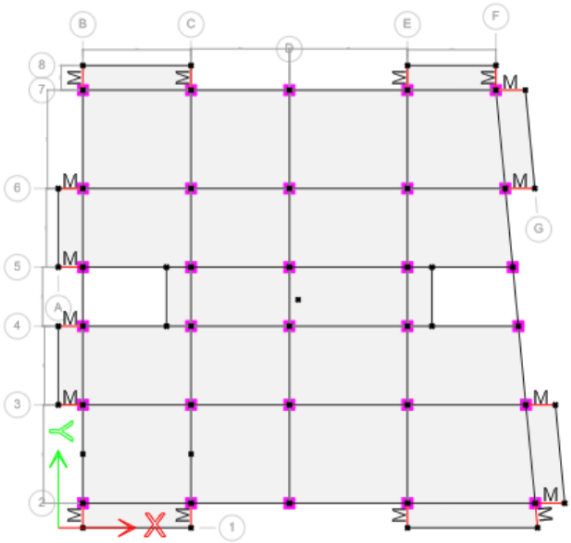
44.10. las vigas en rojo son porque no chequearon y se les debe cambiar de dimensiones.

44.11. Se puede cambiar la dimensión sin quitar el candado, pero se debería haber creado las secciones antes de correr.

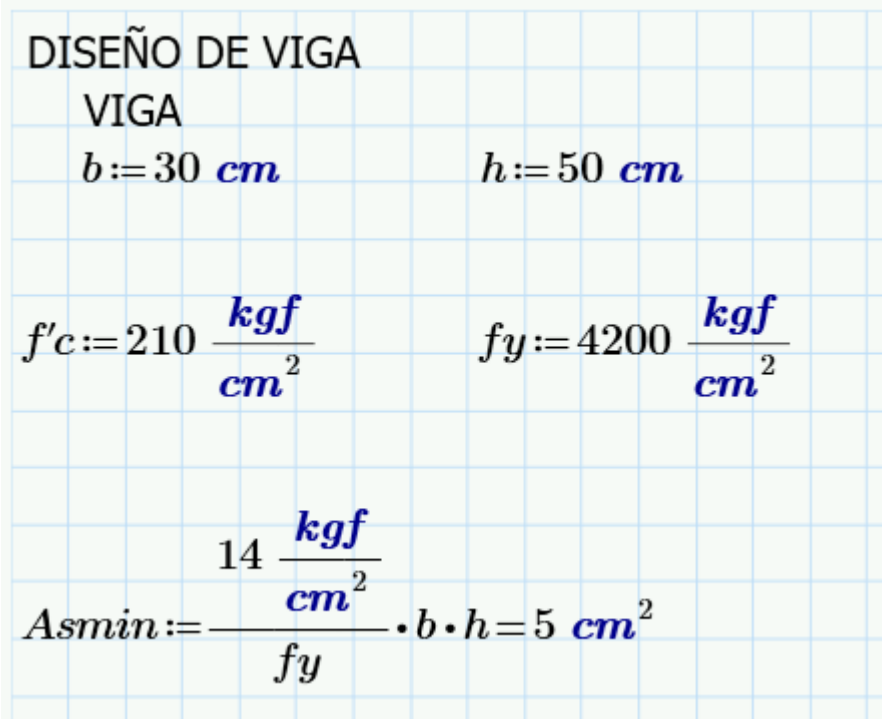


44.12. Para saber el motivo de falla de la viga:





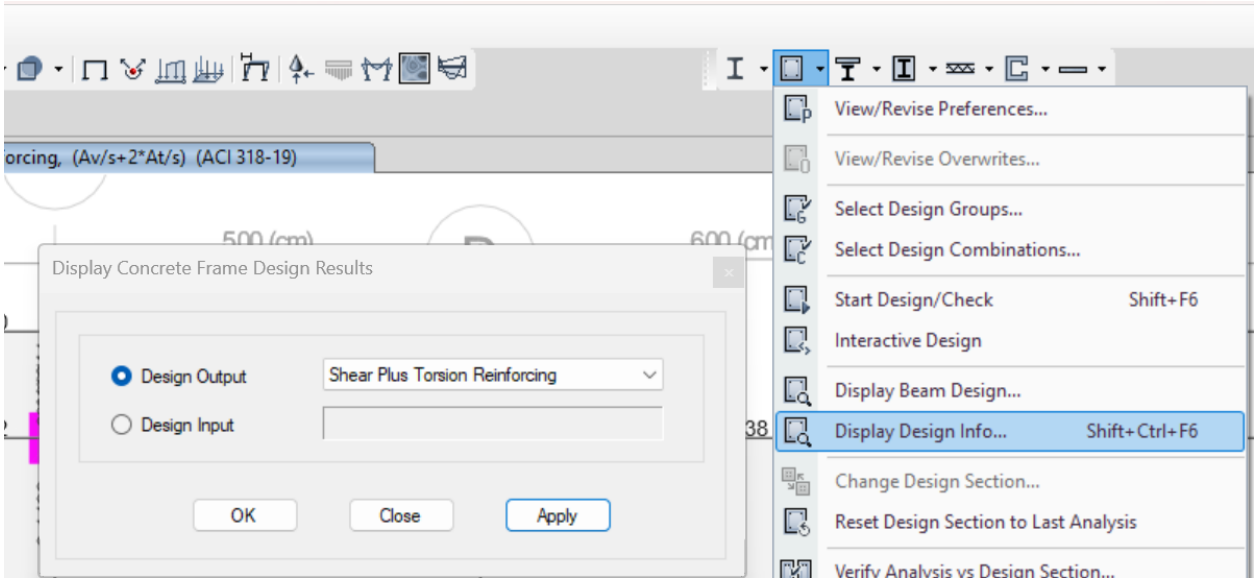
44.13. Ya modificado las dimensiones de las vigas se procede a verificar el Asmin que requiere la viga.



44.14. Se diseña el refuerzo y el bastón

	3No6	14.2511478		9.02572697	
Bastones	3No.4	3.80030609			
	ACERO DEL PROGRAMA				
	13.4		4.4		16.59
	7.5		5		9.56
	AREA DE ACERO REQUERIDO				
	-0.8511478		-9.8511478		2.33885215
	-6.7511478		-9.2511478		-4.6911478

44.15. Se diseña por cortante:



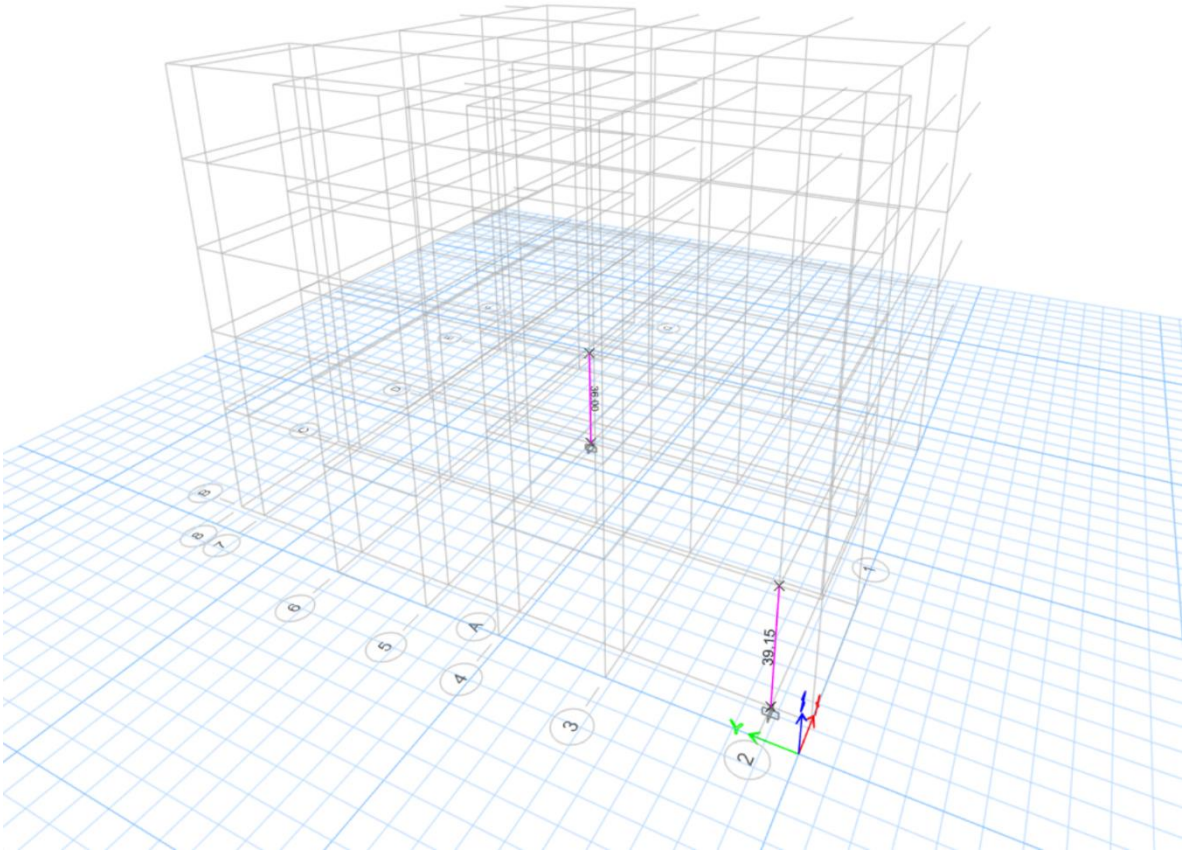
44.16.

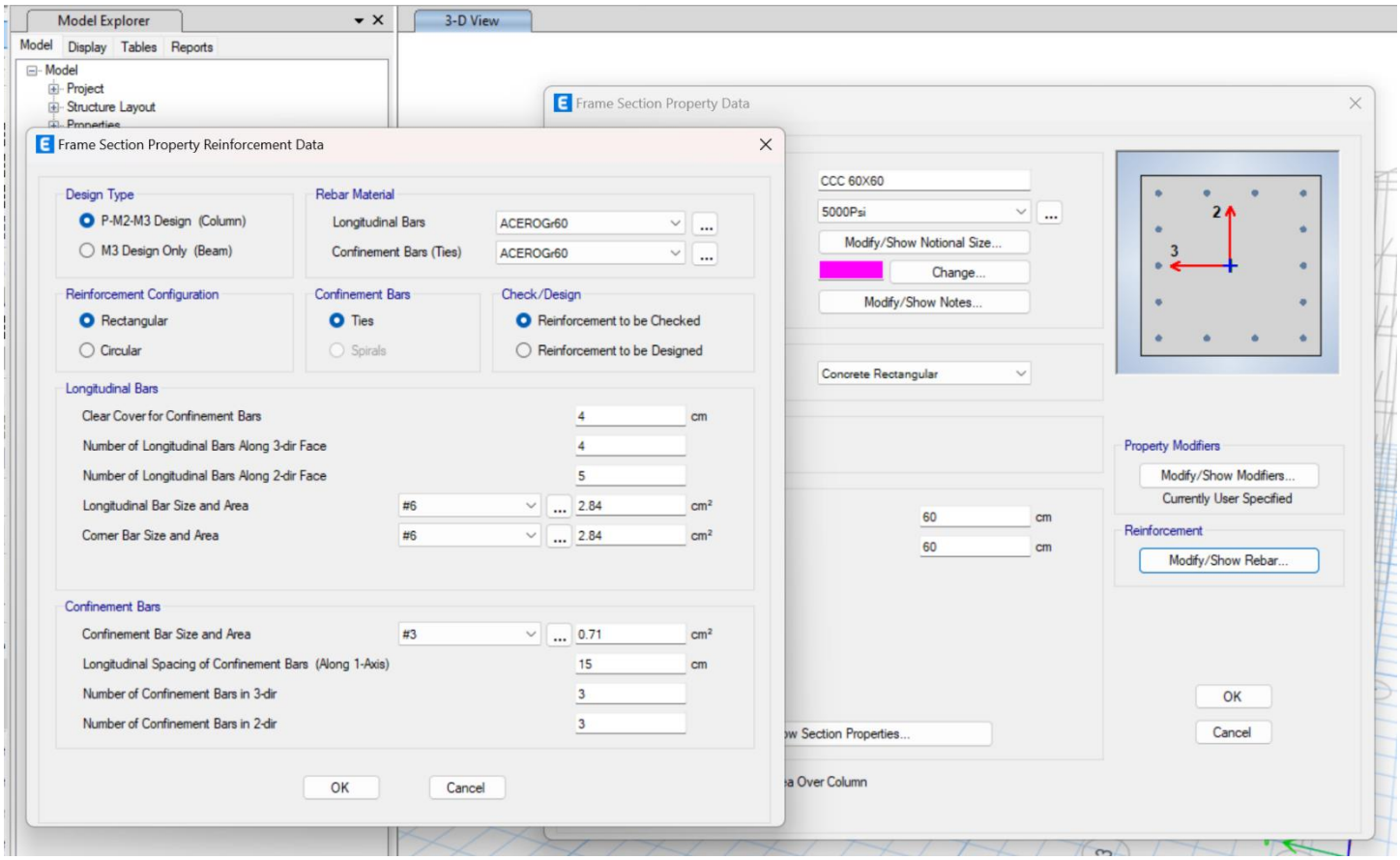
45. **DISEÑO DE COLUMNA.**

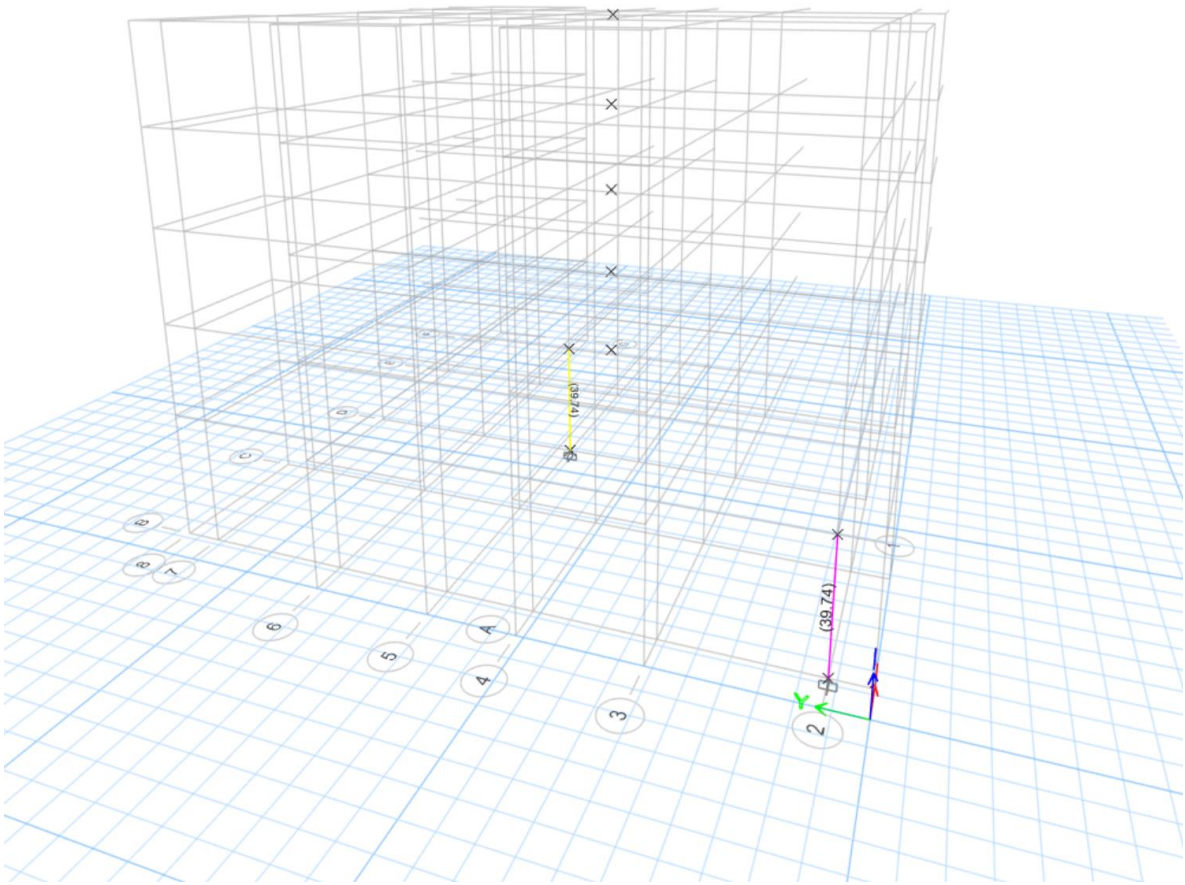
45.1. CORRER EL PROGRAMA Y VER QUE LA CUANTIA DE ACERO ESTE ENTRE EL 1 Y 2%

45.2. CALCULAR LA CANT DE ACERO QUE LLEVARIA LA COLUMNA.

45.3. CREAR UNA COLUMNA CON LAS MISMAS DIMENSIONES PERO QUE CHEQUEEN POR ACERO.

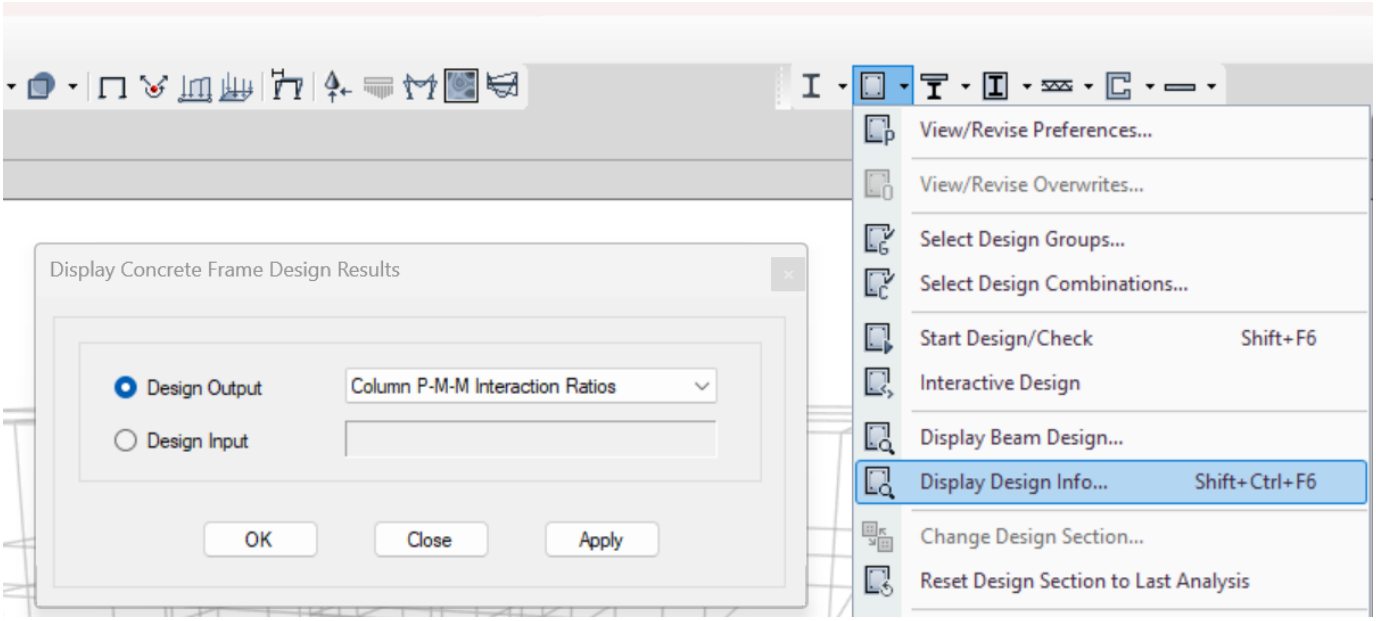


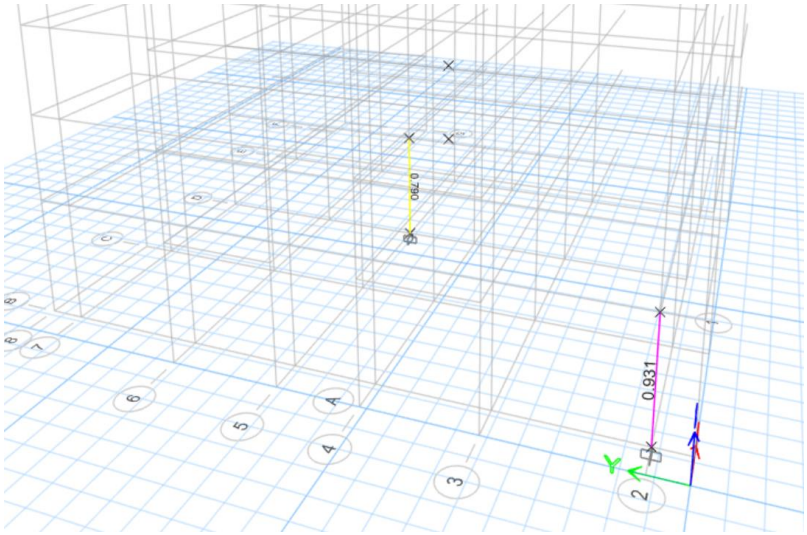

Model Explorer
Model
Display
Tables
Reports
Model
Project
Structure Layout
3-D View
Frame Section Property Data
Frame Section Property Reinforcement Data
Design Type
P-M2-M3 Design (Column)
M3 Design Only (Beam)
Rebar Material
Longitudinal Bars
ACEROGr60
Confinement Bars (Ties)
ACEROGr60
Reinforcement Configuration
Rectangular
Circular
Confinement Bars
Ties
Spirals
Check/Design
Reinforcement to be Checked
Reinforcement to be Designed
Longitudinal Bars
Clear Cover for Confinement Bars
4
cm
Number of Longitudinal Bars Along 3-dir Face
4
Number of Longitudinal Bars Along 2-dir Face
5
Longitudinal Bar Size and Area
#6
2.84
cm²
Corner Bar Size and Area
#6
2.84
cm²
Confinement Bars
Confinement Bar Size and Area
#3
0.71
cm²
Longitudinal Spacing of Confinement Bars (Along 1-Axis)
15
cm
Number of Confinement Bars in 3-dir
3
Number of Confinement Bars in 2-dir
3
OK
Cancel
CCC 60X60
5000Psi
Modify/Show Notional Size...
Change...
Modify/Show Notes...
Concrete Rectangular
Property Modifiers
Modify/Show Modifiers...
Currently User Specified
Reinforcement
Modify/Show Rebar...
60
cm
60
cm
OK
Cancel
Section Properties...
Over Column


(39.74)
(39.74)

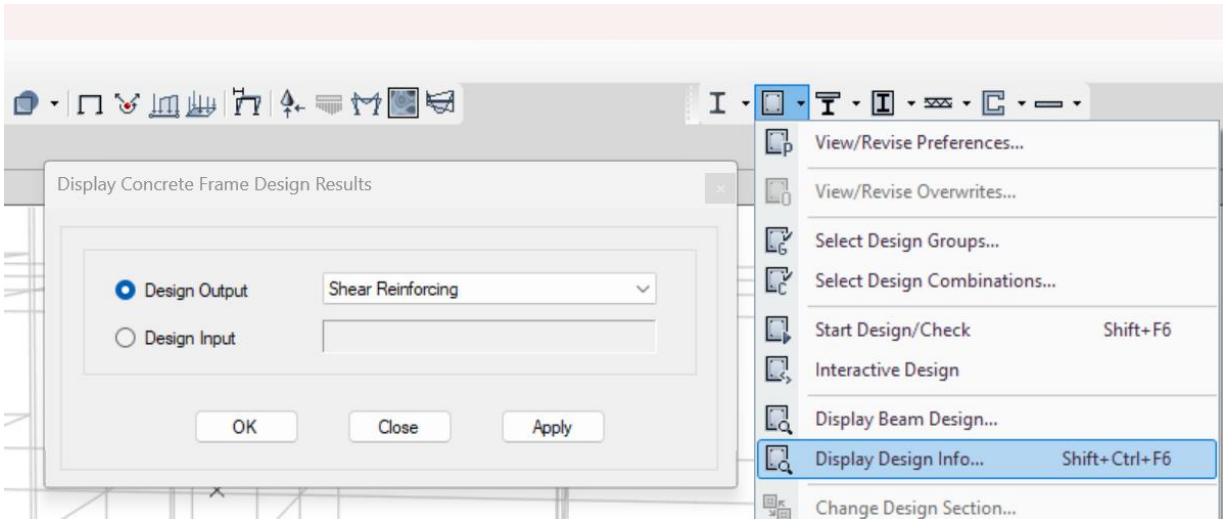
45.4. Chequear la interacción de la columna que se están chequeando por acero

45.5. Para que muestre el porcentaje de acero





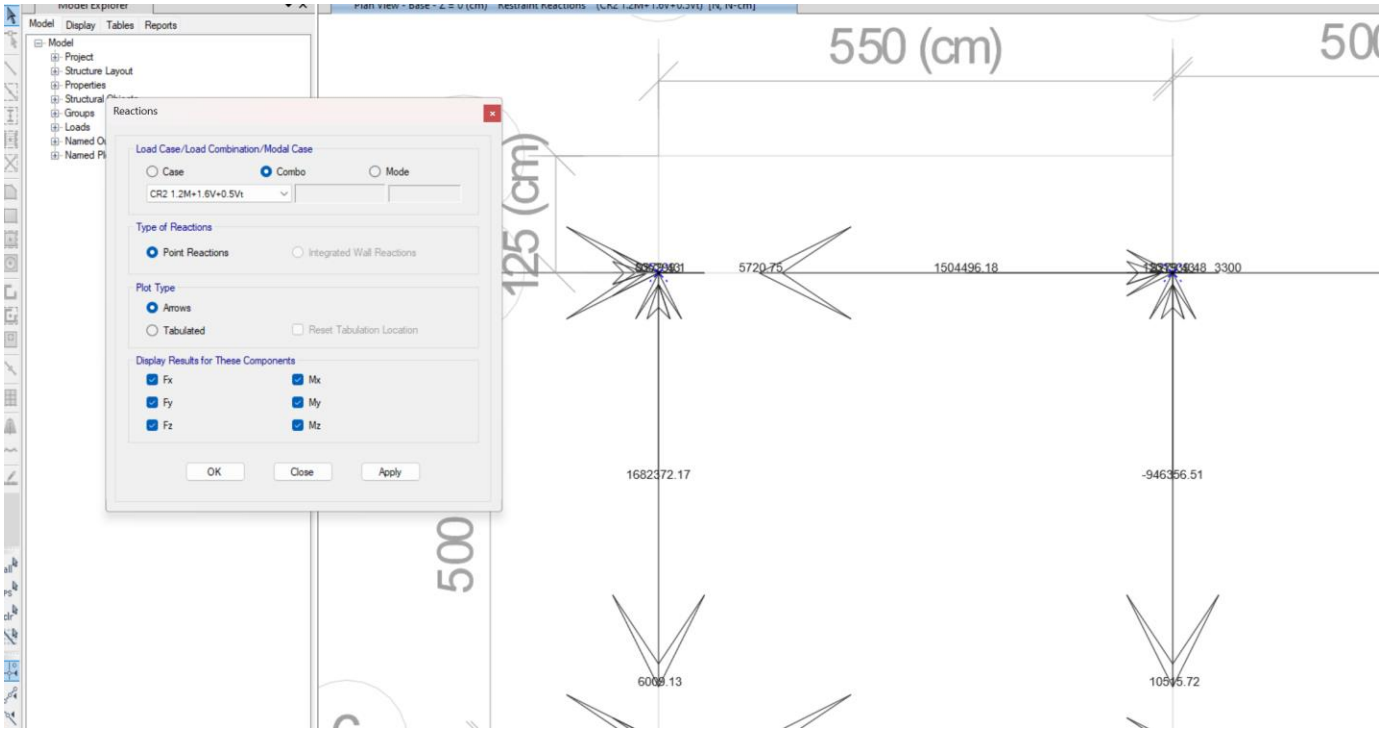
45.6. Obtenemos el acero por cortante.



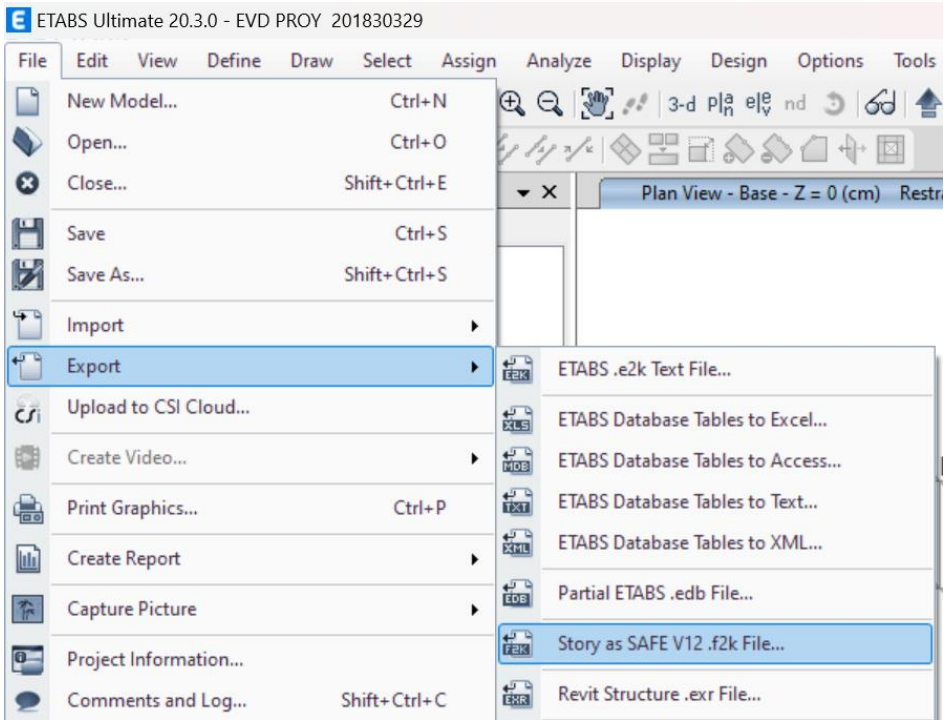
45.7. Luego se hace el análisis por cortante para saber la separación de los estribos en zona confinada y no confinada. (Excel)

46. **DISEÑO DE ZAPATA.**

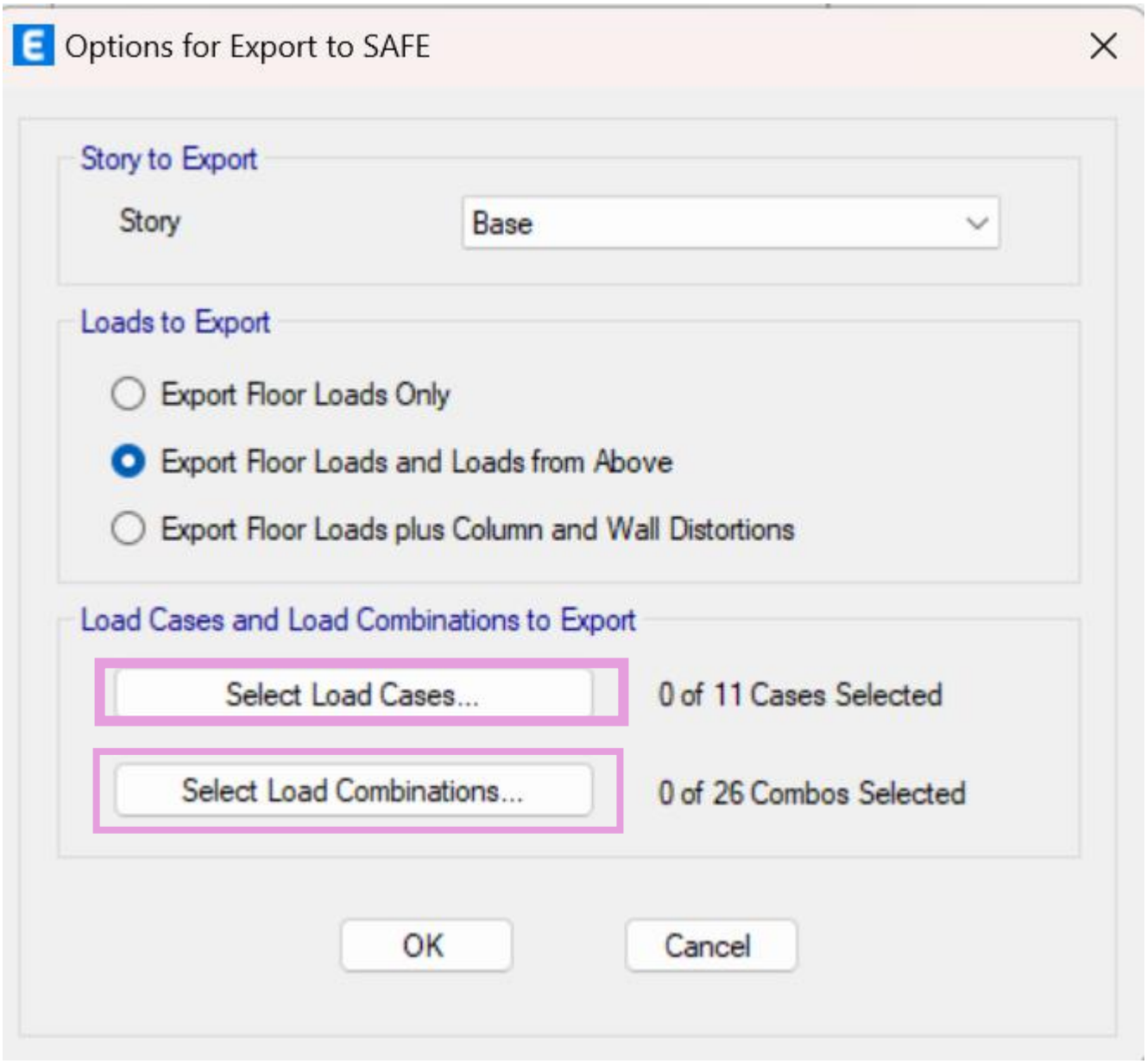
46.1. Para ver las reacciones en los nodos.



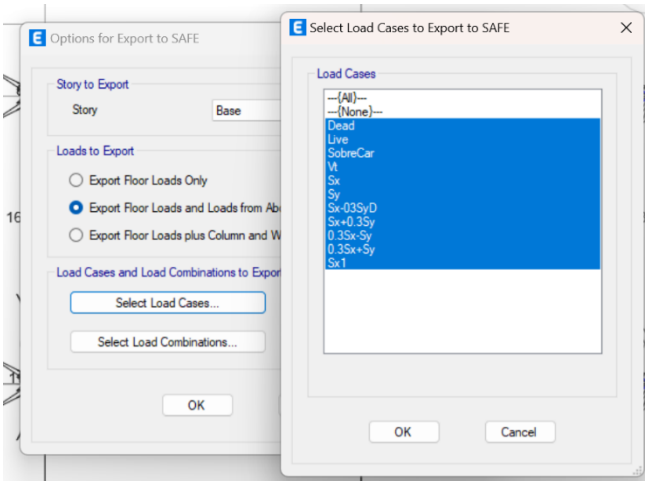
46.2. File > Exportar > Story as SAFE V12

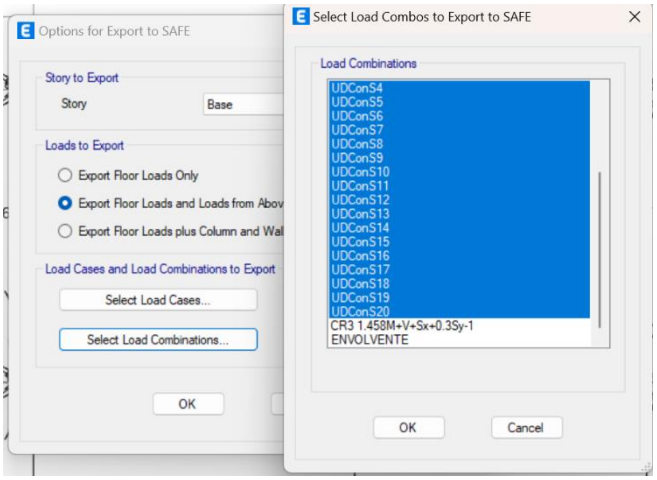


46.3. Se selecciona la 2da opción.

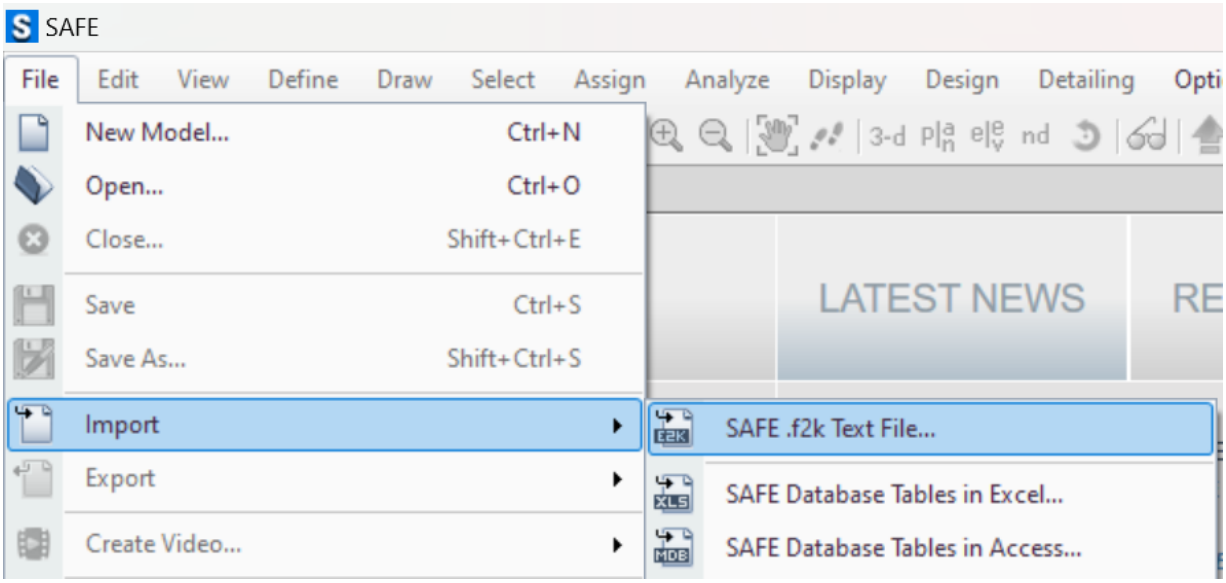


46.4. En combos se selecciona las gravitacionales hasta la UDCON20

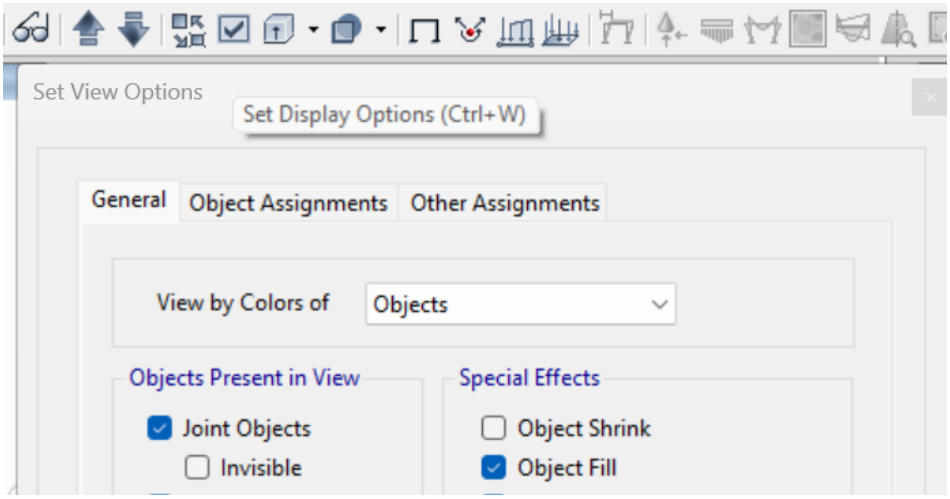




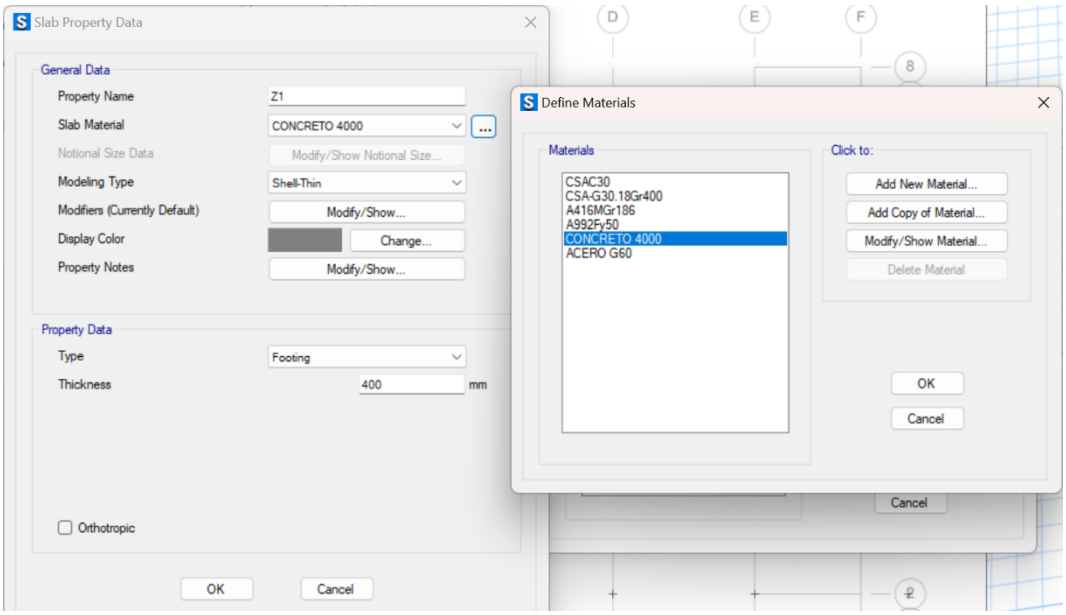
46.5. IR A SAFE E IMPORTAR



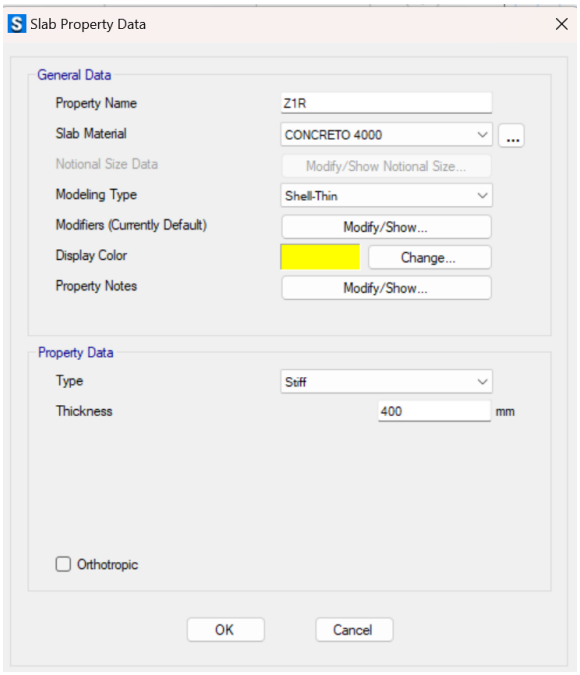
46.6. Quitarle el check a invisible para que aparezcan los puntos.



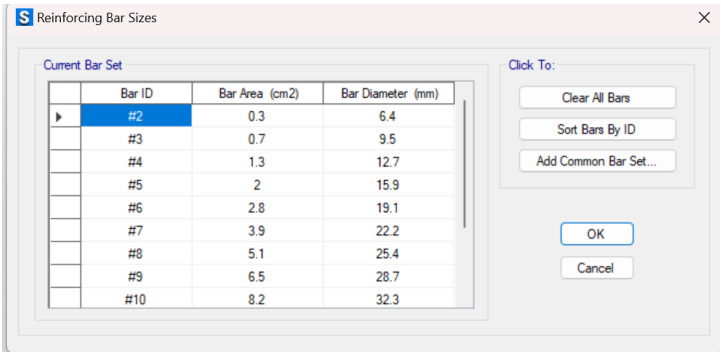
46.7. Se crea un slab y ahí mismo se crean dos materiales de concreto 4000 y acero G60.



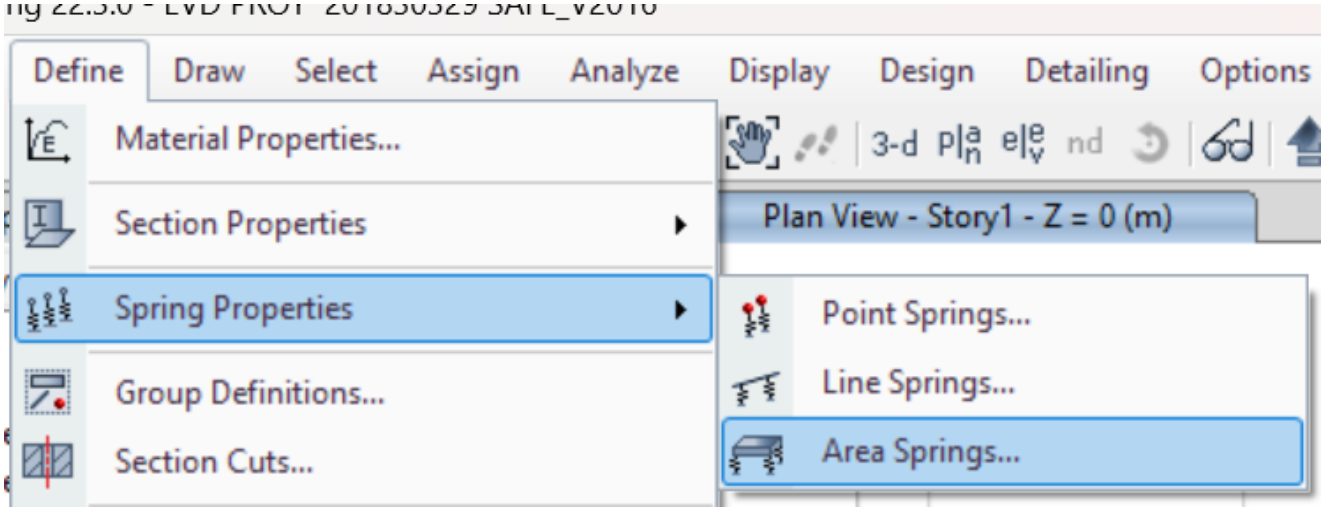
46.8. Luego se crea otra copia pero con diferentes propiedades.



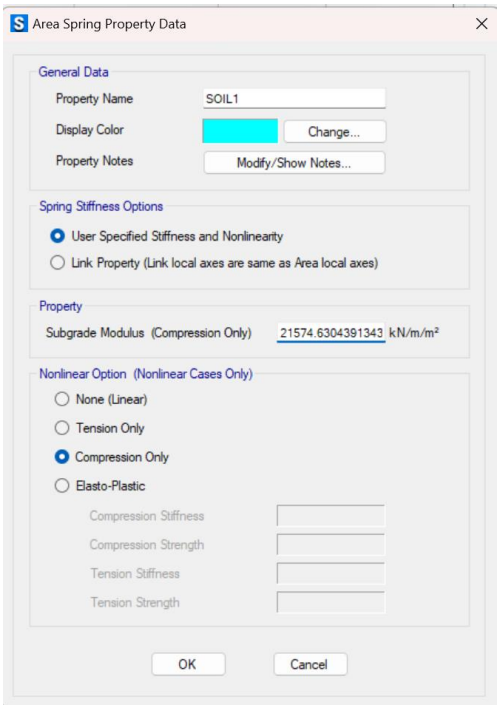
46.9. Limpiar el acero de refuerzo y ponerlos con números.



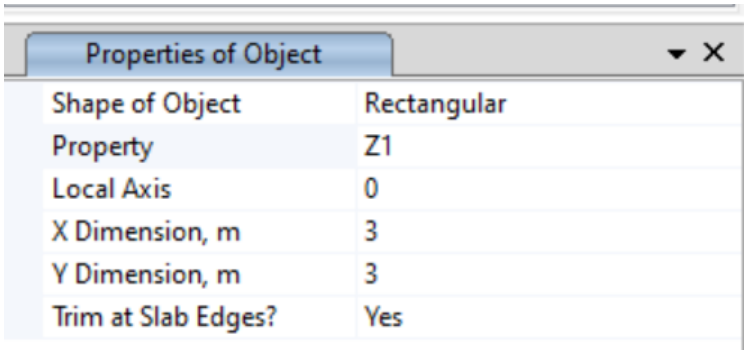
46.10. Verificar las propiedades del suelo.

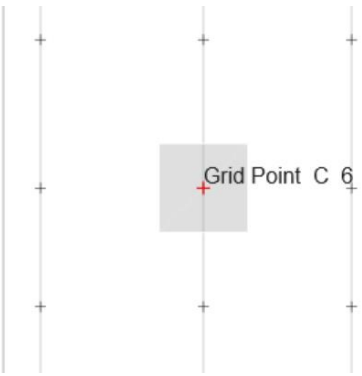


46.11. Se modifica solo



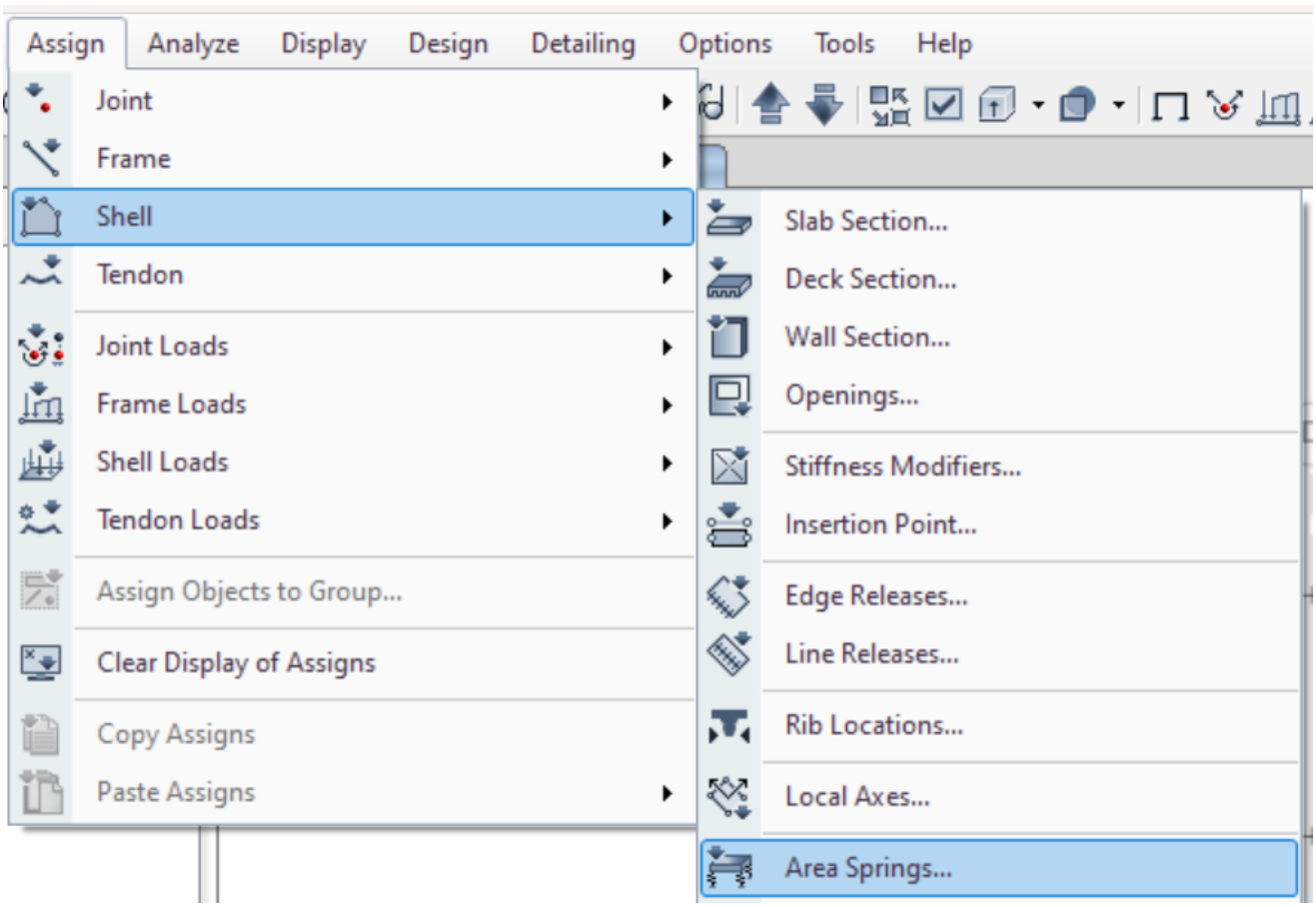
46.12. Dibujar la zapata luego  darl click sobre el punto que queremos dibujar.



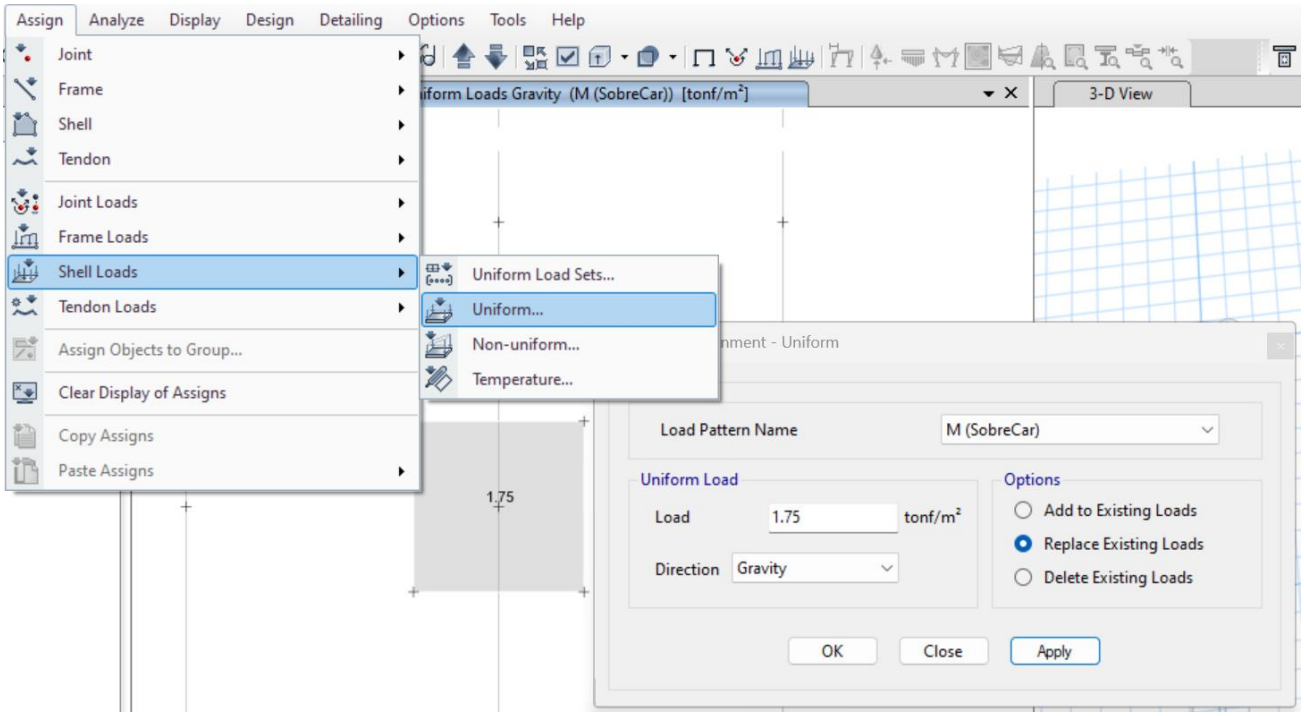


46.13. Asignarle la propiedad del suelo.

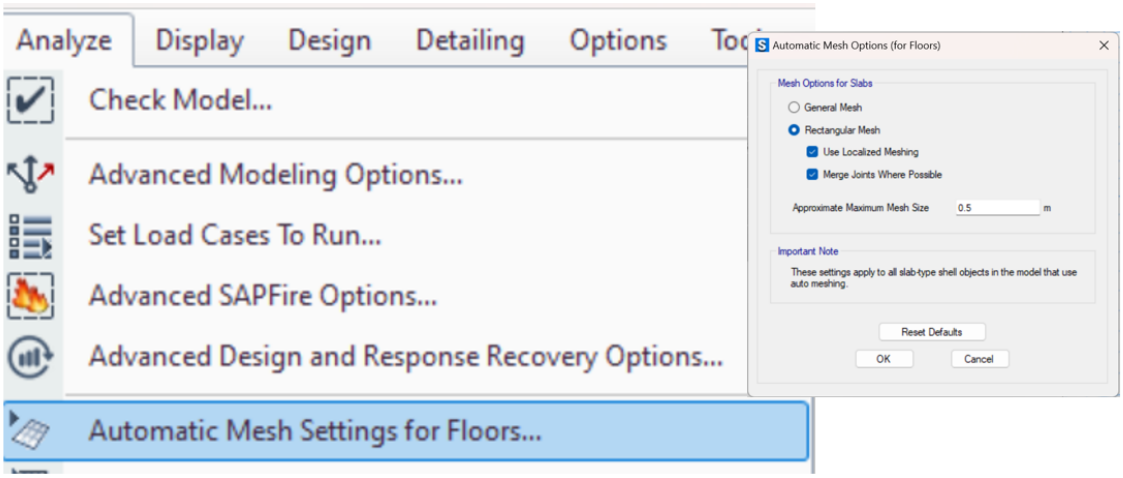
Seleccionar la zapata.



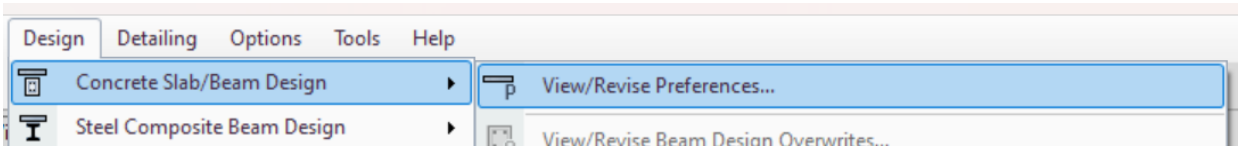
46.14. Asignarle una carga uniforme a la losa (cálculos del Excel)

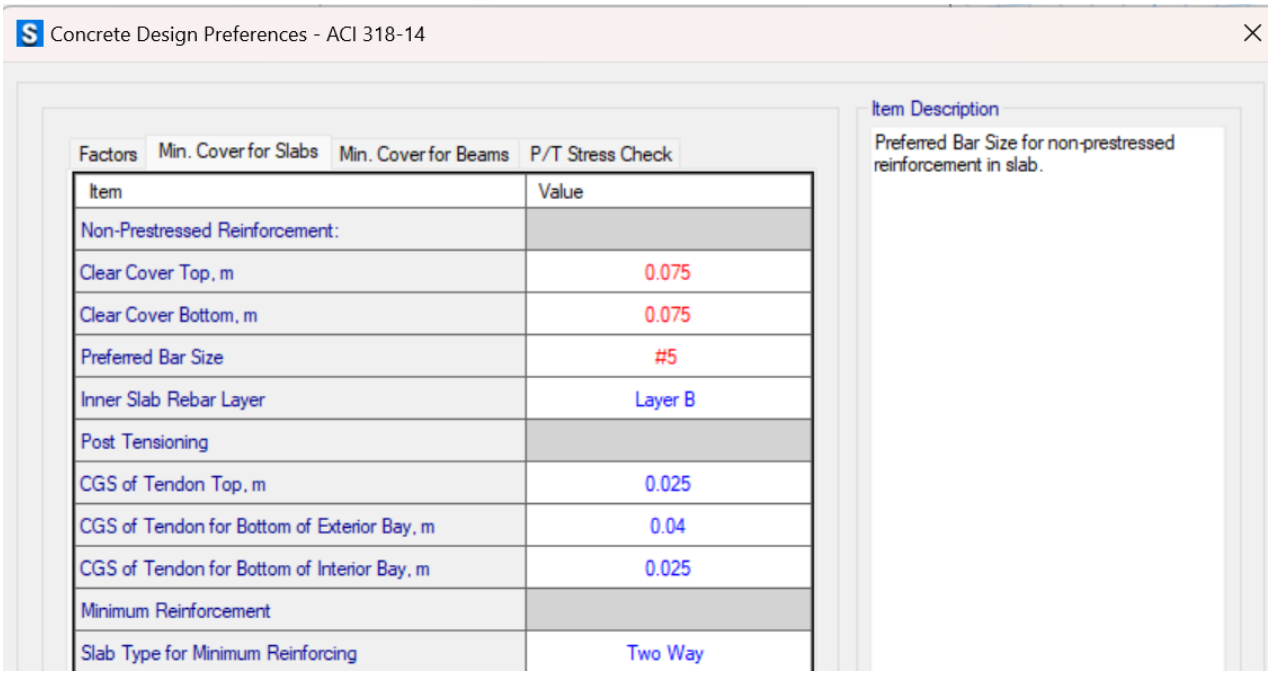


46.15. Asignarle mallado.

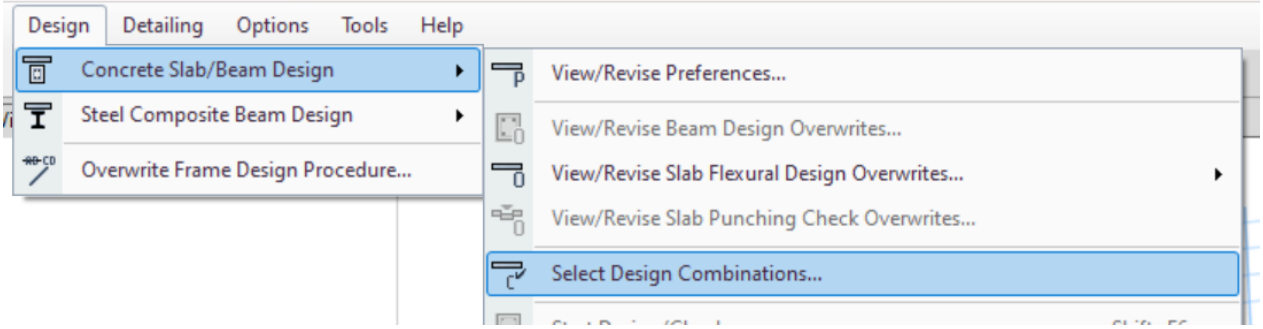


46.16. Modificar preferencias de diseño.

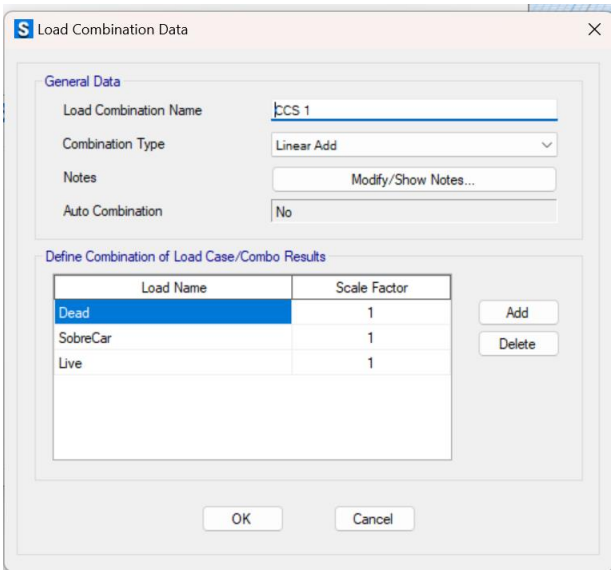


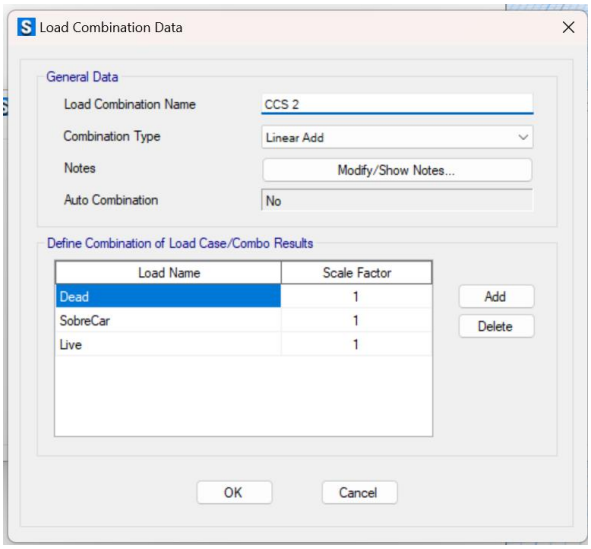


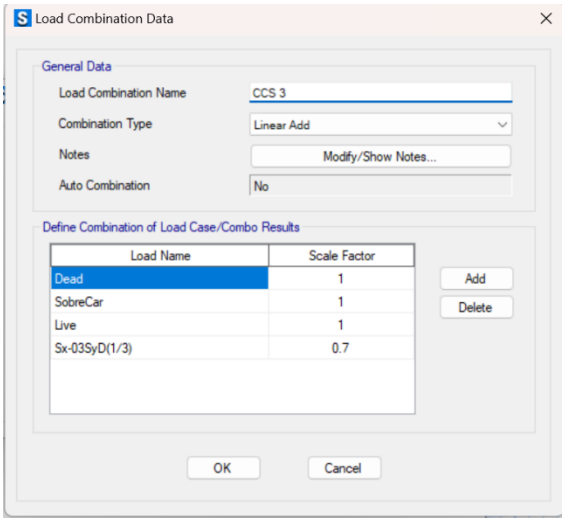
46.17. Pasar las combinaciones de carga al otro lado.

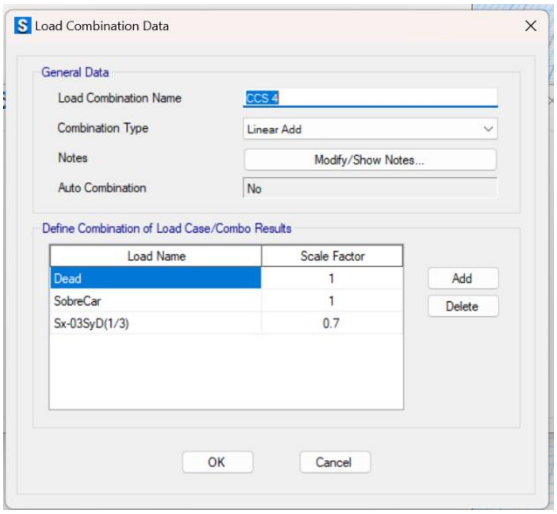


46.18. Agregar Combinaciones de Servicio.



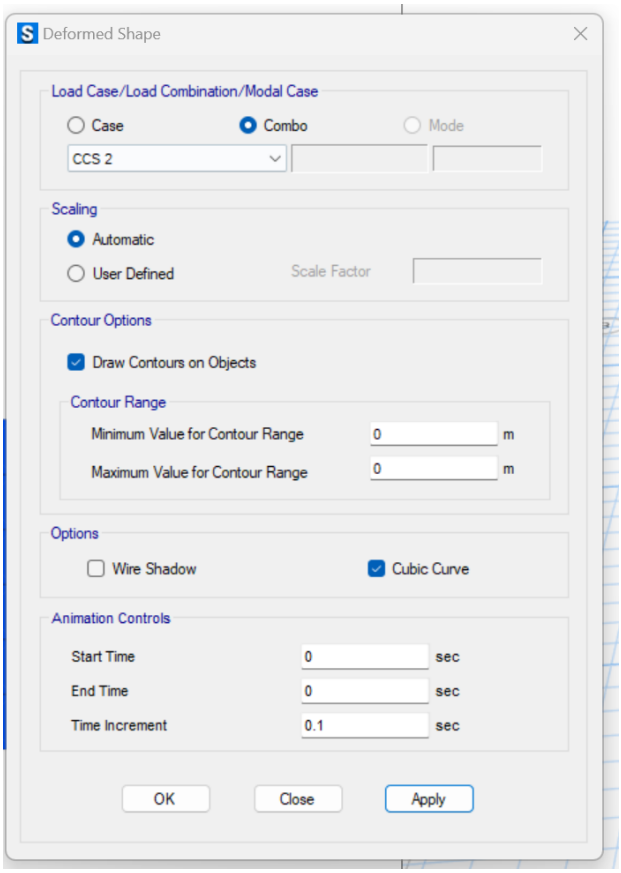




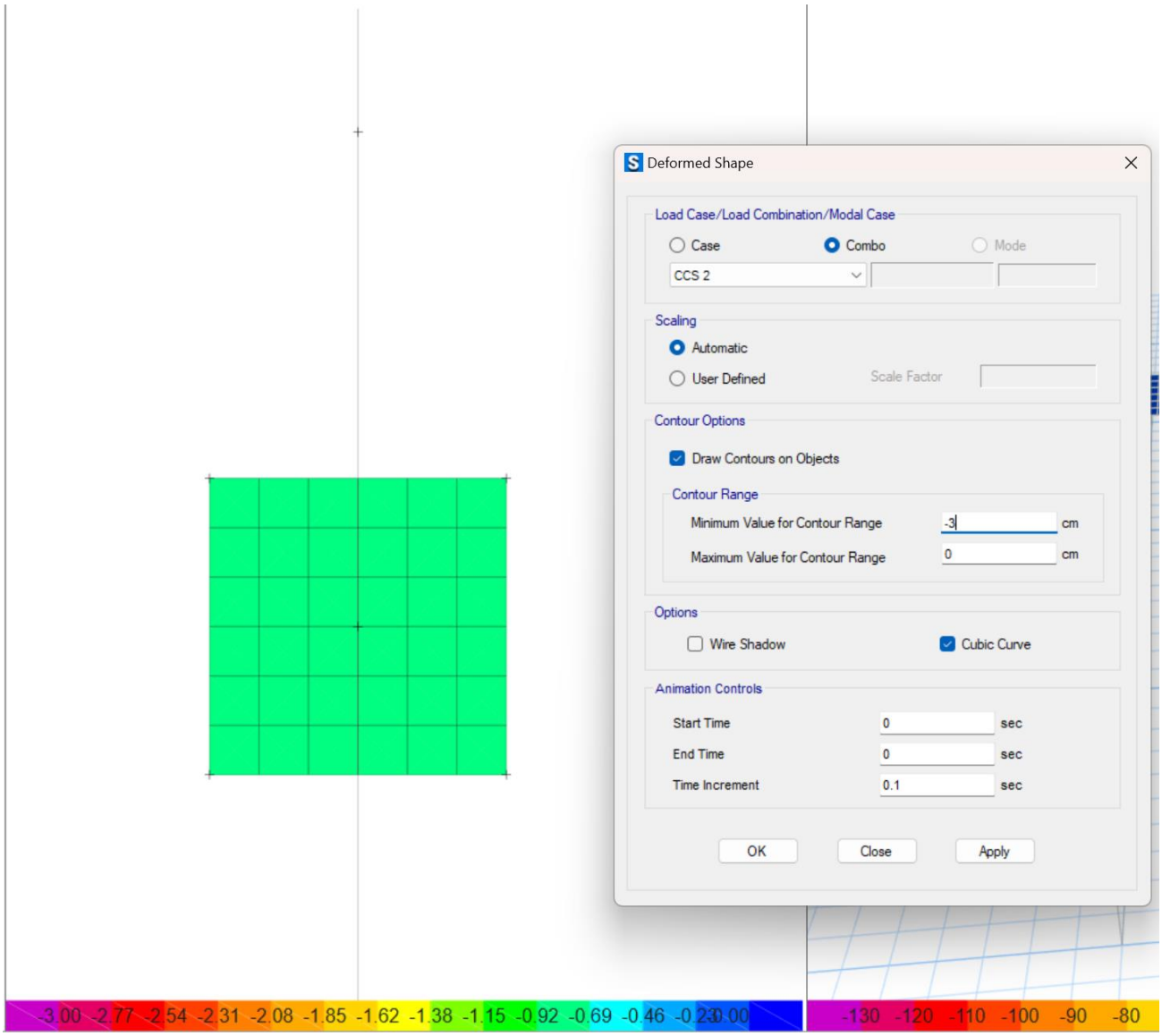


46.19. CORRER EL PROGRAMA

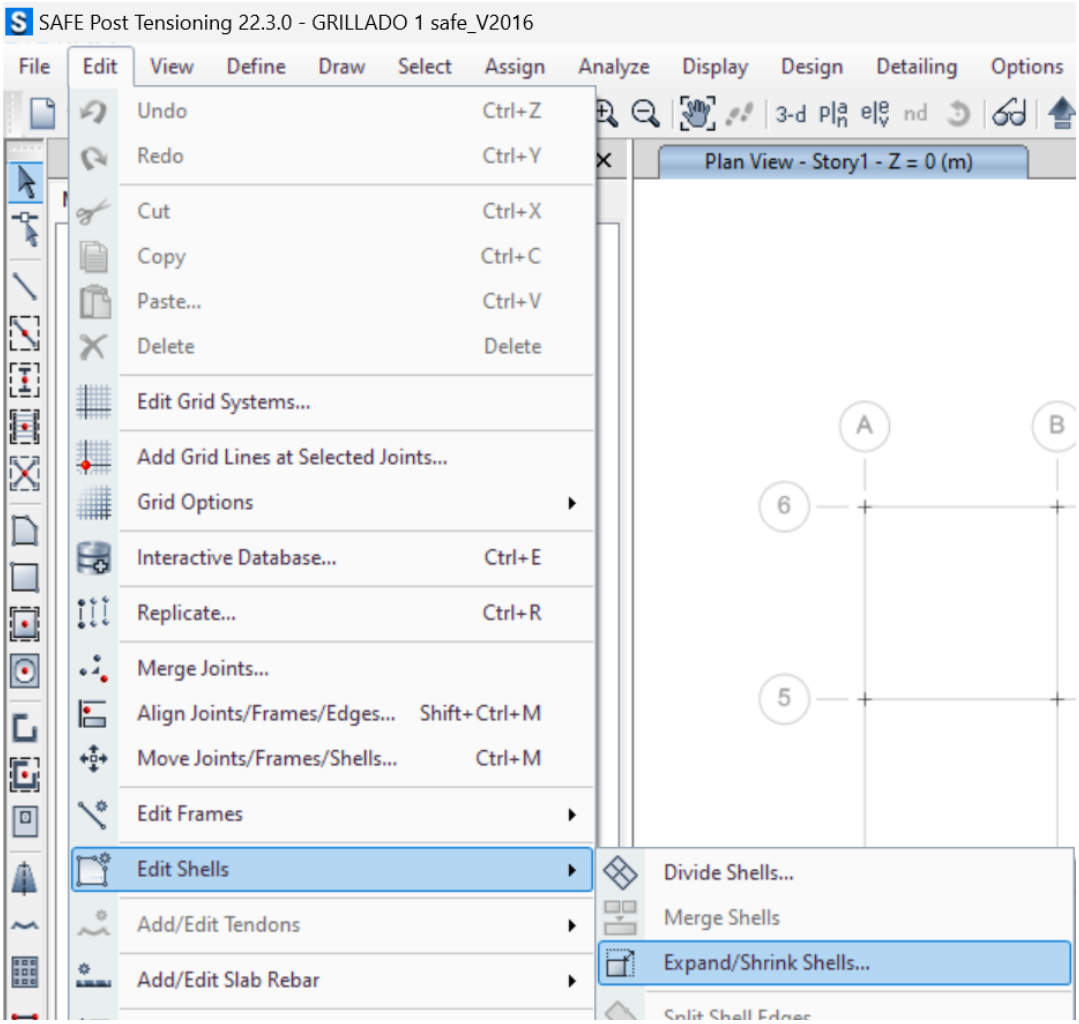
46.20. Obtener la deformada para verificar asentamientos con las CCS 1 y CCS 2.



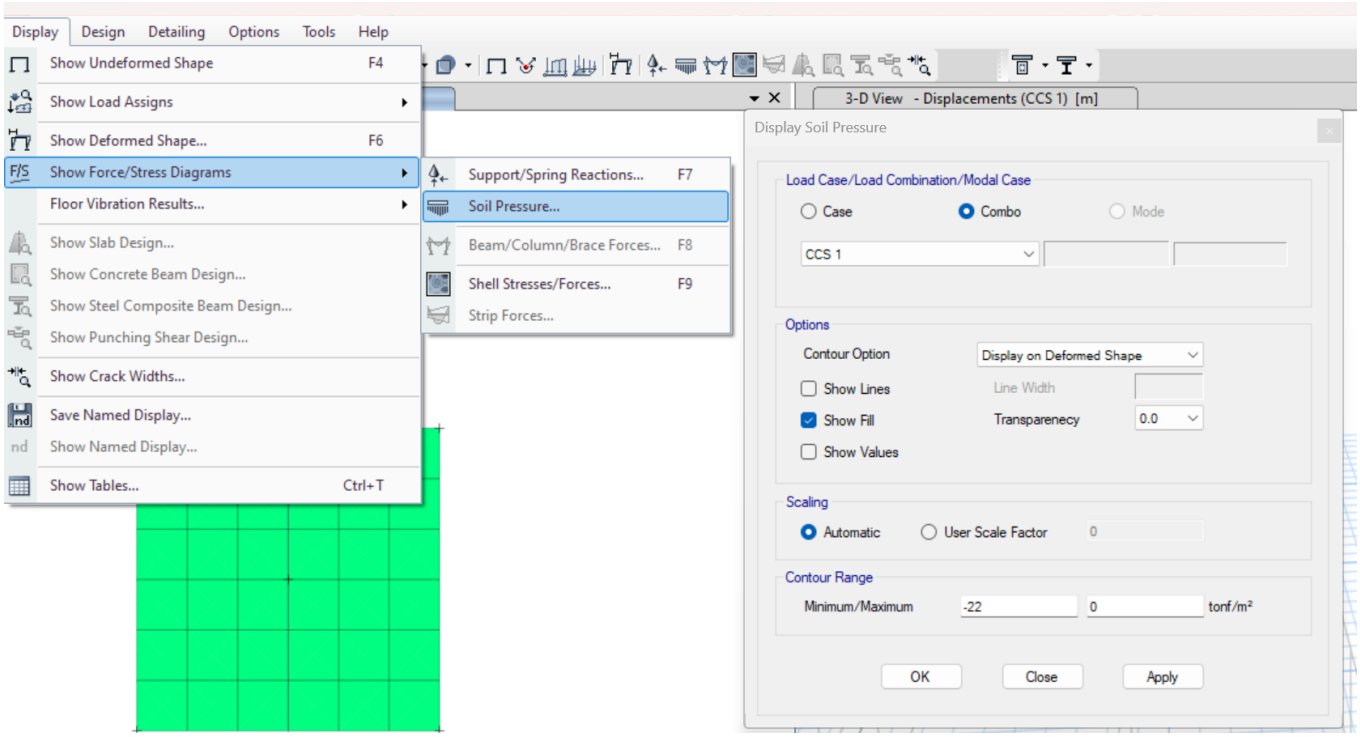
46.21. Ingresar el asentamiento mas y verificar con la franja de colores



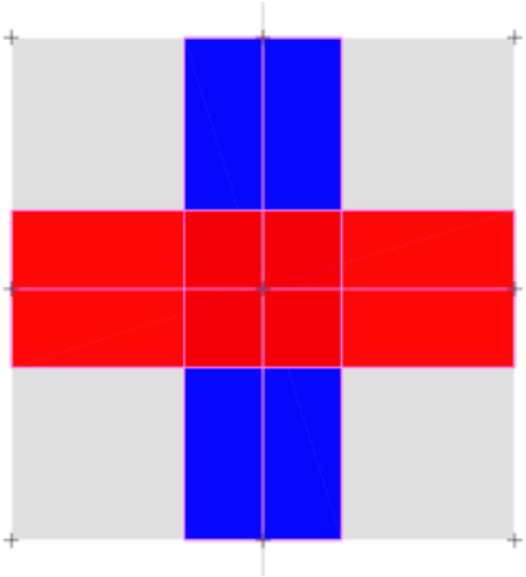
46.22. SI no chequea se cambia de dimensión de la zapata.



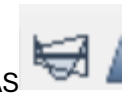
46.23. Chequear la tensión.

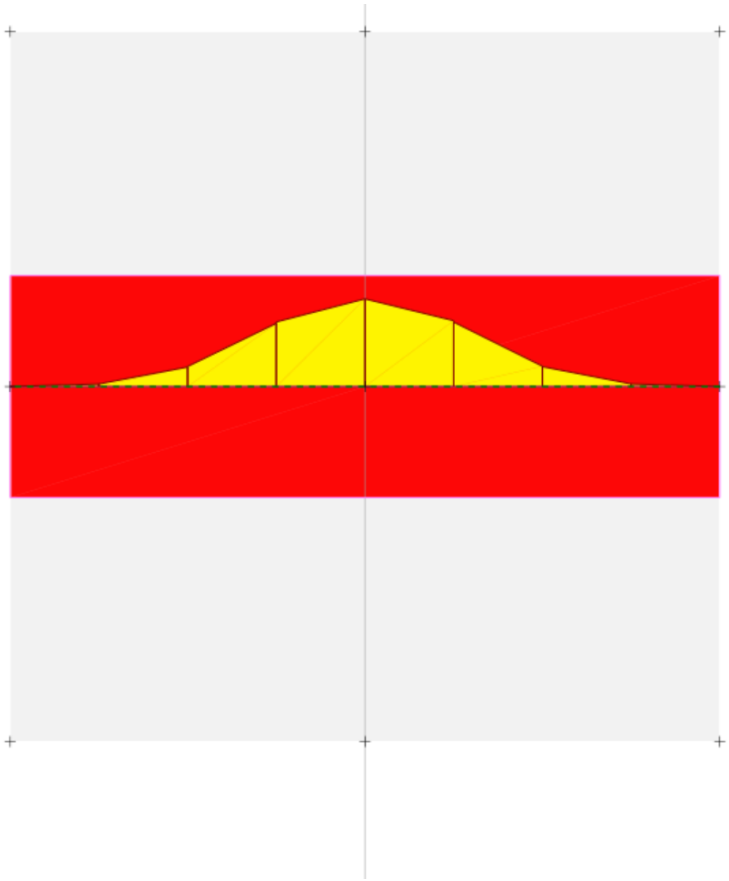


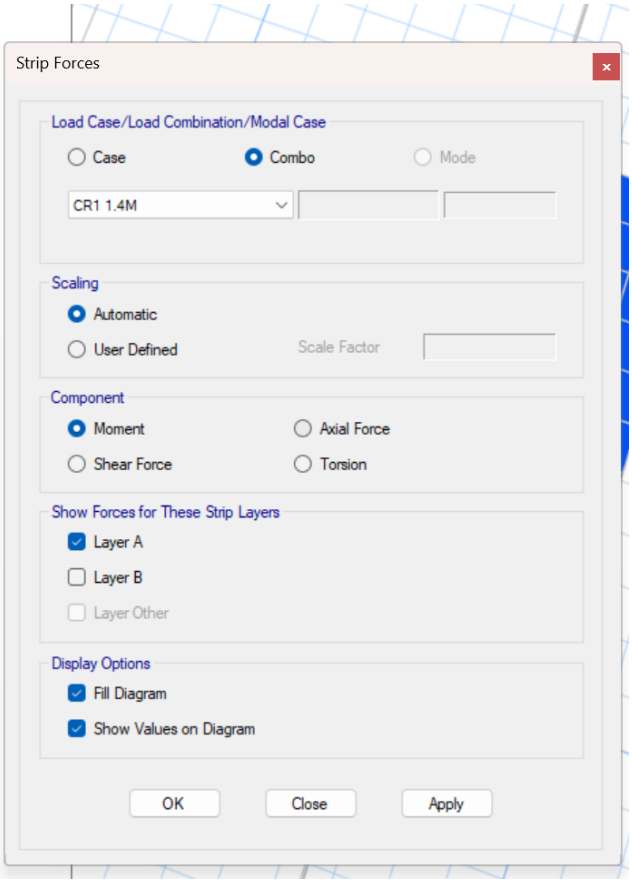
46.24. DIBUJARLE FRANJAS UNITARIAS A LA ZAPATA.



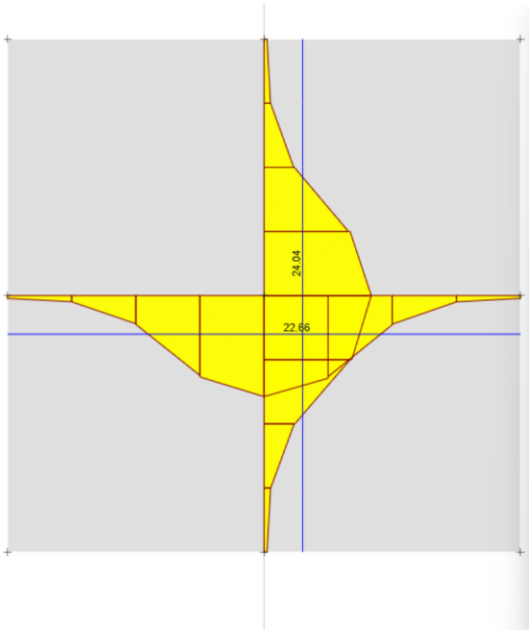
46.25. CORRER EL PROGRAMA.

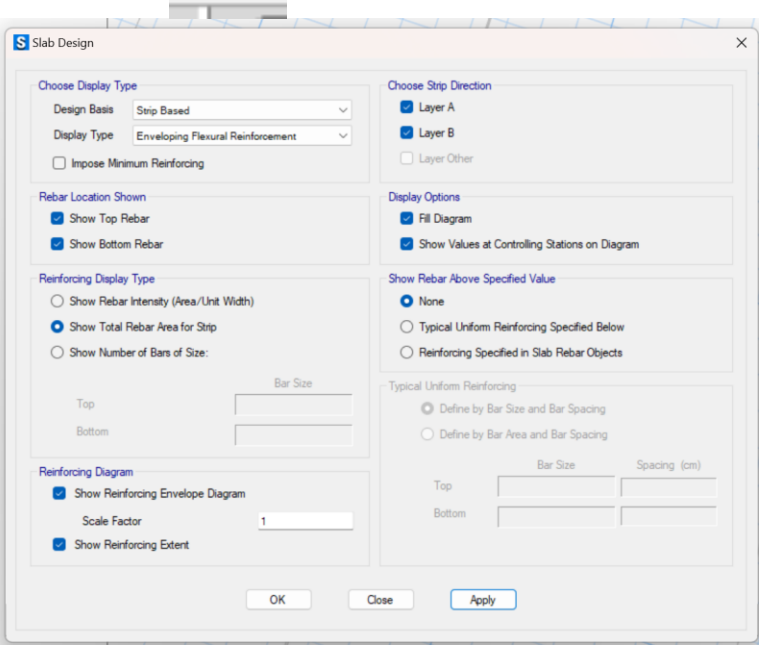
46.26. MOSTRAR LAS FUERZAS DE LAS FRANJAS UNITARIAS 





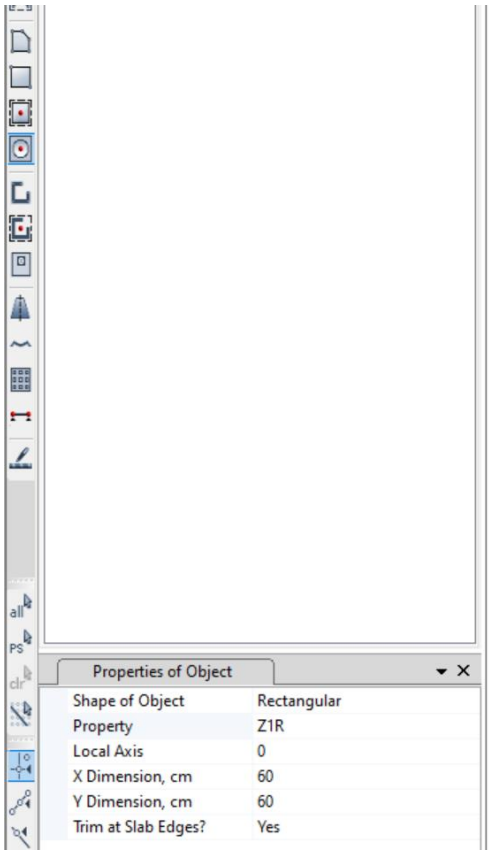
46.27. DISEÑAR EL ARMADO DE LA ZAPATA 

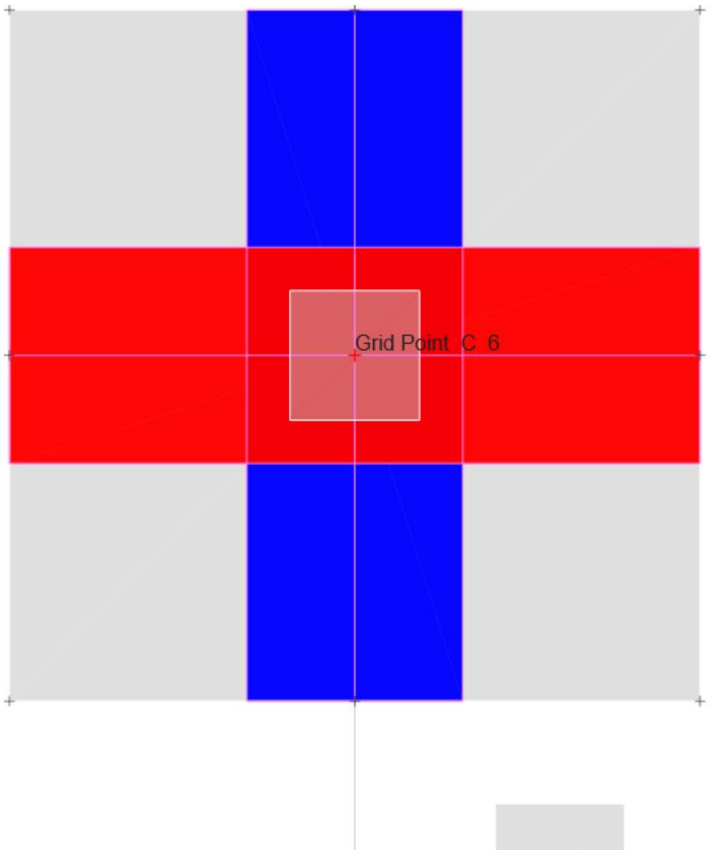




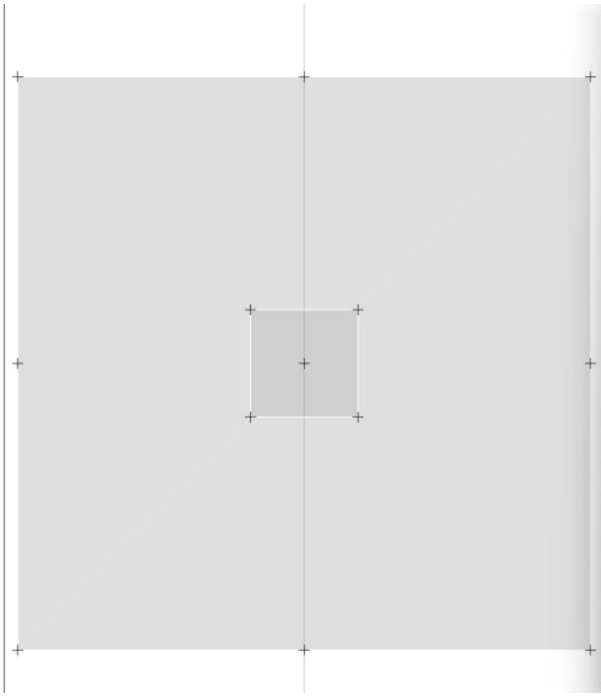
46.28. SE LE AGREGA EL DADO A LA ZAPATA. DIBUJANDOLO COMO UNA ZAPATA NORMAL.

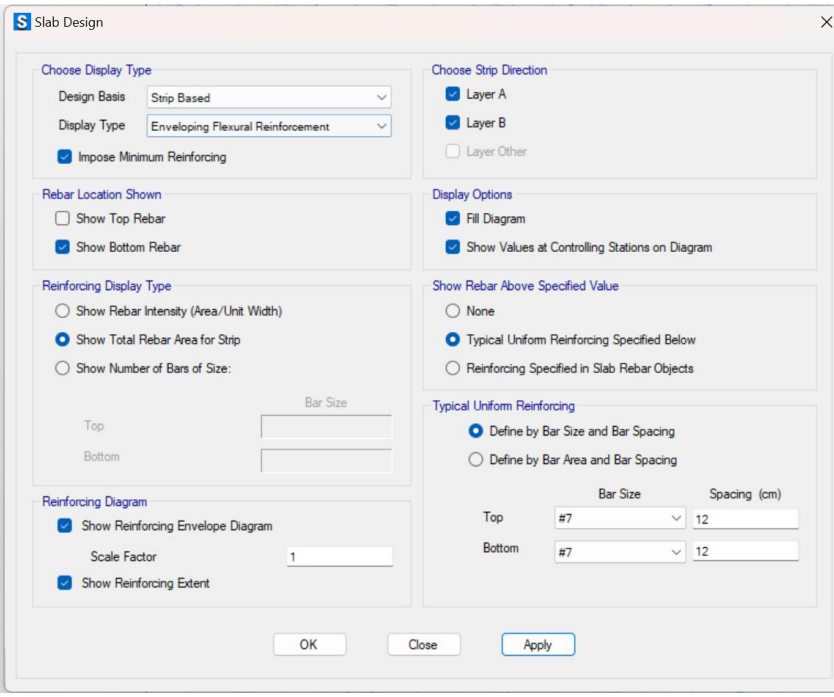
46.29. Y SE LE DA LAS DIMENSIONES DE LA COLUMNA.

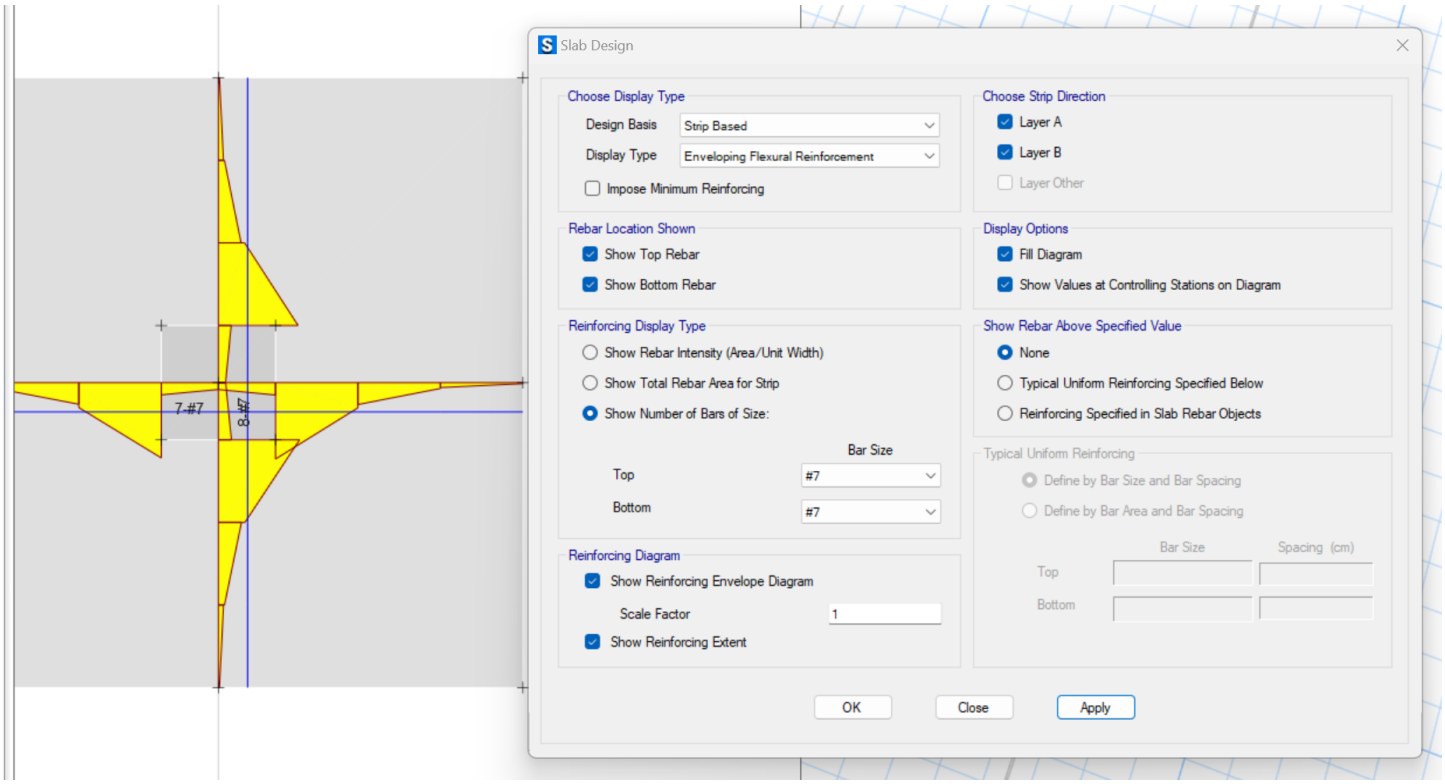




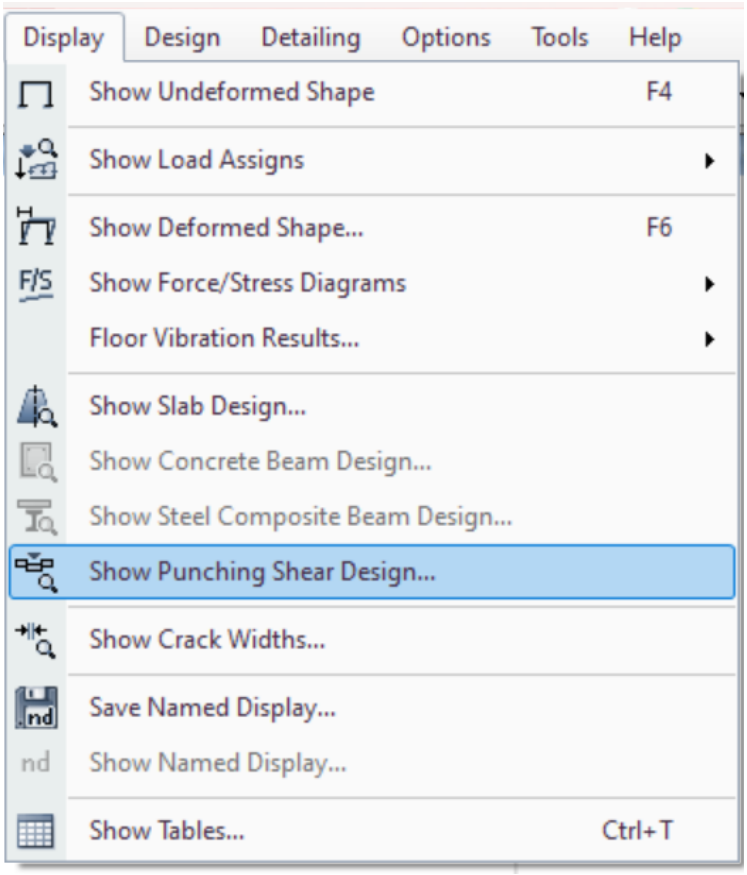
46.30. SE CORRE EL PROGRAMA Y SE VUELVE A CHEQUEAR.







46.31. **VERIFICAR EL PUNZONAMIENTO DE LA ZAPATA.**



ESTE VALOR DEBE SER MENOR A 1, SIDA MAS DE UNO MODIFICAR EL ESPESOR DE LA ZAPATA.

47. **F**
48. **HFH**
49. **FH**
50. **FH**
51. **FH**
52. **F**
53. **H**