Atir Software Development LTD

STRAP - Seismic Analysis

Step by step

All rights, including those of translation, are reserved.

No portion of this document may be reproduced, including photocopying without a written permission from Atir Engineering software LTD.

© Atir Engineering software LTD 13 Kehilat saloniki, Tel Aviv, Israel Tel: +972-3-6480129 Website: <u>http://www.atir.com</u> Email: <u>atir@atirsoft.com</u>

Table of contents

1.	Abstract	. 3
2.	Geometry Definition	. 3
3.	Loads Definition	. 6
4.	Modal Analysis	.7
5.	Seismic Analysis	. 9
6.	Results	15

1. Abstract

The aim of seismic analysis is to calculate the reaction of the model to earthquakes. This example describes the method to analyze the building using Modal Analysis. Modal Analysis is the accepted method and is recommended by mode design codes. The reaction of the model for each mode shape is calculated according to the response spectrum given by the seismic code. Because the aim of this example is to describe the method to do Modal Analysis, we will use a simple six-story structure, with a story height of 3 meters. The seismic load carrying system consists of four shear walls.

2. Geometry Definition

We will define the model geometry and part of the loads using the AutoSTRAP program.



DXF Drawing units m O DXF Units meter O I Program Unit is DXF Units Program units Length: meter Section: cm Material: CDNC Select DXF layers Material: CDNC No. Drawing Layer Assign To 1 COLUMNS Do not load 2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours BF Arrino Dr * load 41 .cN7 .cont load 42 PEN8 Do not load 43 WALLS Walls 44 WALLS Walls 45 NETLIST Do not load 45 NETBRLST Do not load 47 NETBRLST Do not load 47 NETBRLST Do not load 47 NETBRLST Do not load 48 WALL Walls Material 49 NETUST Do not load Material <th></th> <th></th>		
DXF Drawing units DXF Units DXF Units DXF Units DXF Units Program Unit is DXF Units Program units Length: Tenter Section: Cm Material: CONC Select DXF layers No. Drawing Layer Assign To COLUMNS Do not load CONTOUR Contours TEXCOL Do not load RIBS Do not load Contours Textor BF^**** Do not load A Contours Textor BF^**** Do not load A Contours Textor BF^**** Do not load A Contours Textor Contours C	~	
● DXF Units cm ✓ ● 1 Program Unit is DXF Units Program units Eength: meter Force: Ton Section: cm ✓ Material: CDNC Select DXF layers Material: CDNC No. Drawing Layer Assign To 1 1 COLUMNS Do not load 2 2 TEXCOL Do not load 3 3 RIBS Do not load 4 4 CONTOUR Contours - - BF^**** Do not load 4 41 c.N7 uo not load - 42 PEN8 Do not load - 43 WALLS Walls - 44 WALLS Walls - 45 NETLIST Do not load - 46 TEXNTLST Do not load - 47 NETBRLST Do not load - • the elements identified by the program are element grid, click on the S icon. - • click on	~	
○ 1 Program Unit is DXF Units Program units Force: Length: meter Section: cm Contours CONC Select DXF layers No. Drawing Layer Assign To 1 COLUMNS 2 TEXCOL 0 not load 2 TEXCOL 0 not load 3 RIBS 0 not load 4 CONTOUR Contours - BF**** 0 r 42 PEN8 0 not load 43 WALLS 44 WALLHASH 0 not load 45 NETLIST 0 not load 46 TEXNTLST 0K Cancel • the elements identified by the program are element grid, click on the side me box: • click on • Levels • in the side me box:	~	
Program units meter Force: Ton Section: om Material: CONC Select DXF layers Assign To 1 No. Drawing Layer Assign To 1 1 COLUMNS Do not load 2 2 TEXCOL Do not load 3 3 RIBS Do not load 4 4 CONTOUR Contours 4 - BF^**** Do not load 4 41 .cN7 .oo not load 4 42 PEN8 Do not load 4 43 WALLS Walls 4 44 WALLHASH Do not load 4 45 NETLIST Do not load 4 46 TEXNTLST Do not load 4 47 NETBRLST Do not load 4 • the elements identified by the program are element grid, click on the si icon. • • click on tevels in the side me box: Add levels X Y Y Y	~	
Length: meter Force: Ton Section: cm Material: CONC Select DXF layers Assign To 1 No. Drawing Layer Assign To 1 1 COLUMNS Do not load 2 2 TEXCOL Do not load 3 3 RIBS Do not load 4 4 CONTOUR Contours - - BF**** Do not load 4 41 .cN7 Jo not load 4 43 WALLS Walls - 44 WALLHASH Do not load 4 45 NETLIST Do not load 4 46 TEXNTLST Do not load 4 47 NETBRLST Do not load 4 • the elements identified by the program are element grid, click on the icon. • click on - Levels in the side me box: Add levels X Y Levels In the side me box:	~	
Section: cm Material: CONC Select DXF layers Assign To 1 No. Drawing Layer Assign To 1 COLUMNS Do not load 2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours BF^**** Dr<* load	~	
Select DXF layers No. Drawing Layer Assign To 1 COLUMNS 2 TEXCOL 3 RIBS 4 CONTOUR Contours BF***** BF***** Do not load 41 Assign To 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST DK Cancel • the elements identified by the program are element grid, click on the side me box: Add levels Add levels Number of levels to add :	~	
Select DXF layers Assign To 1 COLUMNS Do not load 2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours - BF^***** Do not load 41 .cN7 Do not load 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXINTLST Do not load 47 NETBRLST Do not load 48 Cancel Cancel		
No. Drawing Layer Assign To 1 COLUMNS Do not load 2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours - BF^***** Dr 41 .cN7 .ro not load 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load 48 element grid, click on the in the side me box: Add levels × Creatilev Add levels × Creatilev Number of levels to add : 6		
1 COLUMNS Do not load 2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours - BF^···· Dr * load 41 .cN7 .roo not load 42 PEN8 Do not load 43 WALLS Walls 44 WALLS Walls 45 NETLIST Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load • the elements identified by the program are element grid, click on the load in the side me box: Add levels × Createl Number of levels to add : 6	^	
2 TEXCOL Do not load 3 RIBS Do not load 4 CONTOUR Contours - BF^····· Do not load 41 . cN7 Do not load 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load 48 Cancel Cancel		
3 RIBS Do not load 4 CONTOUR Contours - BF^····· Dr ···load 41 . cN7	Click Here and	
4 CONTOUR Contours - BF^**** Dr **load 41 . cN7 Jo not load 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load 48 WALLS Cancel • the elements identified by the program are element grid, click on the side me box: in the side me box: Add levels X Creat levels Number of levels to add : 6	from the list	
- BFATTA Dristionad 41 . cN7 Joinot load 42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load 48 Cancel Image: Context of the elements identified by the program are element grid, click on the Sicon. • click on the Levels in the side me box: Image: Creat levels in the side me box: Add levels X Creat levels in the side me box: Number of levels to add : 6 Image: Creat levels in the side me box:		
41 . cN7		
42 PEN8 Do not load 43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load 48 Cancel • the elements identified by the program are element grid, click on the side me box: • click on • Levels in the side me box: Add levels × Created box:	_	
43 WALLS Walls 44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load • the elements identified by the program are element grid, click on the side me box: • click on • Levels in the side me box: Add levels × Created by the program are element of levels to add :		
44 WALLHASH Do not load 45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load • OK Cancel • the elements identified by the program are element grid, click on the ≤ icon. • click on + Levels in the side me box: Add levels X Creat levels Number of levels to add : 6	Click Here and	
45 NETLIST Do not load 46 TEXNTLST Do not load 47 NETBRLST Do not load OK Cancel OK Cancel • the elements identified by the program are element grid, click on the ≤ icon. • click on ← Levels in the side me box: Add levels × Number of levels to add : 6	select "Walls" from	
46 TEXNILST Do not load 47 NETBRLST Do not load 47 NETBRLST Do not load 47 OK Cancel • the elements identified by the program are element grid, click on the ≪ icon. • click on ← Levels in the side me box: Add levels × Creat levels to add : 6	the list	
 INETERLEST DO NOTION OK Cancel the elements identified by the program are element grid, click on the side me box: Add levels X Created by the program are element grid, click on the side me box: 	_	
 OK Cancel the elements identified by the program are element grid, click on the icon. click on Levels in the side me box: Add levels X Creater levels Number of levels to add : 6 	_	
 OK Cancel the elements identified by the program are element grid, click on the icon. click on Levels in the side me box: Add levels × Creater levels Number of levels to add : 6 	>	
 OK Cancel the elements identified by the program are element grid, click on the icon. click on Levels in the side me box: Add levels × Createlevels Number of levels to add : 6 		
 the elements identified by the program are element grid, click on the icon. click on Levels in the side me box: Add levels × Creater levels Number of levels to add : 6 		
 the elements identified by the program are element grid, click on the icon. click on Levels in the side me box: Add levels × Create levels Number of levels to add : 6 		
element grid, click on the sicon.	displayed on the	screen. To display
 click on Levels in the side me box: Add levels × Created levels Number of levels to add : 6 		
 click on the side me box: Add levels Number of levels to add : 		
box: Add levels × Crea Number of levels to add : 6	nu and Add Level a	at the bottom of t
Add levels X Crea Number of levels to add : 6		
Add levels X Created levels		
Number of levels to add : 6	te six	
Number of levels to add : 6	els	
Level beight :		
With	a spacing of 3	
OK Cancel	ers between	
	ach level	
	ach level	

• The program adds the list of levels to the Dialog box. click <i>End</i> to continue.
 Click Loads in the lower side menu, then select Slab load in the upper side menu.
Click Select all spaces and define the loads:
Slab Loads
Loads: Dead: 0.5 Ton/m^2 Live: 0.3 Ton/m^2 Self Weight factor: 1 Not Loaded
 Do not change Apply loads as : Element loads Beam loads - bidirectional Beam loads - unidirectional Horizontal Vertical
O Angle OK Cancel
 Create the STRAP model: click STRAP in the bottom side menu. Click Defaults in the top side menu and check Create rigid links in plane. This option creates infinite rigidity in the plane of the floor slab.
Click Create model in the top side menu Strap Model Selection ×
Model Name: Seismic Analysis Project Folder: D:\Tutorials Overwrite existing model A submodel Oreate a plane model will be created at OK Cancel
Seismic Analysis 5



STRAP

- Run STRAP and open the created model Seismic Analysis.
- click 💷 in the icon bar to display an isometric view.
- select the submodel to display from the list in the side menu:
- click 🖾 in the icon bar to display the walls.
- define "dummy beams" on the slab perimeter (for applying line loads to the slab): click
 Beams
 Beams
 in the lower side menu, then click
 Define
 in the upper side menu.
- Set the following two parameters:
- \circ Prop. = 0 in the bottom dialog box (indicates that the beams are 'dummy').
- \circ \blacksquare Chain with previous beam in the options at the top of the side menu.
- Select the four corner nodes in sequence, then select the first node again to end the beam definition..
- display the Main model: Main model

3. Loads Definition

Add the walls self-weight and the masonry line loads along the slab perimeter (the slab self-weight was defined as part of the slab dead load):

- click the Loads tab.
- select Existing load in the side menu, select the "Dead" load case and click
 Revise load
- select Wall loads in the lower side menu and select Define in the upper side menu.
- click on Self weight, select the local x3 direction and define a factor = -1.0.
- click on *Select all walls*; the load is applied to all the walls in the main model.
- select the first instance of the submodel: EX4#1
- select Beam loads in the lower side menu and select Define in the upper side menu.

		ING SOFTWARE DEVELOF
	click on Uniform, then Select all beams and define the loads:	
	Beam no. 225 L= 0.5 Specify the load direction	
	X3 Direction: FX3	
	Start End Tune: Global	
	Load= 1 ton/m Specify the	
	Apply load as joint load	
	Apply load to Apply to all	
	C Selected instances of the submodel submodel instances	
	OK Cancel	
	roturn to the Main model	
•	click End load case	
•	click 1+2 ⁵ Solve to solve the model for static loads	
•	click 1+2 ⁵ Solve to solve the model for static loads	
• 4. M	click 1+2 ^{= Solve} to solve the model for static loads	
• 4. M •	click 1*2 ^s Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads.	
• 4. M •	click ^{1+2^s Solve} to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads.	
• 4. M •	click Let's Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option:	
• 4. M •	click ^{1+2^s Solve} to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load	
• 4. M •	click Load <pre>select factors for load cases:</pre> <pre>to solve the model for static loads </pre>	e dead
• 4. M •	click ^{1+2^s} Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name 1 1	e dead oction of I, based
• 4. M •	click 1*2* Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name 1 2 Live	e dead botion of l, based c code lents
• 4. M •	click click the Weights tab. The masses are defined from the static loads. click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name Factor Apply all the load and a p the live load on seismic requirem	e dead ootion of l, based c code ients
• 4. M •	click Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name Factor Apply all the load and a p the live load on seismic requirem	e dead ootion of l, based c code ients
• 4. M •	click ^{1+2*} Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name Factor Apply all the load and a p the live load 0.2 Live 0.2 on seismic requirem	e dead potion of l, based c code tents
• 4. M •	click ^{1+2*} Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name Factor 1 Dead 1. 2 Live 0.2 Apply all the load on seismic requirem	e dead potion of l, based c code lents
• 4. M •	click 1*2* Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name Factor Live 0.2 Apply all the load and a p the live load on seismic requirem Addition mode: Apply option to:	e dead potion of l, based c code lents
• 4. M •	click 1*2° Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the Static load option: Static Load Select factors for load cases: No. Load case name I Dead C Live Addition mode: A	e dead ootion of l, based c code ients
• 4. M •	 click ^{1+2^e} Solve to solve the model for static loads Independent of the static load static load static loads Independent of the static load static load static load to not static loads Independent of the static load static load to not static load Independent of the static load static load to not static load Independent of the static load static load to not static load Independent of the static load static load Independent of the stati	e dead ootion of l, based c code lents
• 4. M •	click ^{1+2^z} Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the static load option: Static Load Select factors for load cases: No. Load case name Factor for load cases: No. Load case name Factor for load cases: No. Load case name Factor for load cases: Apply all the live load on seismic requirem Addition mode : All model nodes Selected nodes Static load component: C X1 C X2 C X3	e dead ootion of l, based c code lents
• 4. M •	click ** ^{2*} Solve to solve the model for static loads Iodal Analysis click the Weights tab. The masses are defined from the static loads. select the static load option: Static Load Select factors for load cases: No. Load case name 1 2 Live 2 Addition mode : Addition mode : Addition mode mights Addition mode : Apply option to : Addition mode : OK Cancel	e dead option of based c code eents
4. M •	click *** solve to solve the model for static loads	e dead oction of based c code eents
4. M •	click ** ^{2*} Solve to solve the model for static loads Iodal Analysis click the weights tab. The masses are defined from the static loads. select the static load option: Static Load Select factors for load cases: No. Load case name Factor for load cases: No. Load case name Factor for load cases: No. Load case name Factor for load cases: Apply all the load on a prive for load case in the live load on a seismic requirem Addition mode: Addition mod	e dead oction of based code ents

•	PTIR I ENGINEERING SOFTWARE
•	No of mode shapes to be calculated = 10
	Calculate natural frequencies within a convergence tolerance of : 10 ⁻³
	Apply the mass
	for horizontal seismic action \checkmark X1 direction \checkmark X2 direction $dx1 = [0.$ $dx2 = [0.$ \checkmark X2 direction $dx2 = [0.$ \checkmark X3 direction
	Calculate soft stories and shear center
	Stories Eccentricities OK Cancel
	click the Eccentricities button to define an earthquake in the X1 direction.
	Set selection Current set : X1 New Delete
	Earthquake direction : X1 Image: Constraint of the second se
	No. Level (m) Height (m) dx1 (m) dx2 (m) dx3 (m) A 0.00 0
	5 15.00 3.00 6 18.00 3.00
	Print
	OK Cancel
	_
	Seismic Analysis

_

				OPMENT LTD
click the	Stories	to verif	fy the levels for Story data calculations.	
Story Definition	on		×	
		Story tolerance :	0.5	
No.	Level (m) 0.00	Height (m) 🔺	In Levels for	
1	3.00	3.00	story data	
2	6.00	3.00	calculations	
3	9.00	3.00 🥌		
4	12.00	3.00		
5	15.00	3.00	Delete	
6	18.00	3.00		
			Print	
			ΟΚ	
<		>	Cancel	
 click ^{1+2[≠]} S 	olve	to calculate	the mode shapes. After the modal analysis is solve	ed, ye

- click _______ to calculate the mode shapes. After the modal analysis is solved, you
 will get into the ________ tab.
- display the results graphically: select a mode shape and then click *Start Animation / End Animation* at the bottom of the screen to start/stop animation; repeat for other mode shapes.

5. Seismic Analysis

• •Select the method for combining the modes: select *Seismic analysis* in the menu bar and *Method for combining modes* in the pulldown menu:



• **SRSS**: square root of sum of squares. The estimated response R (force, displacement, etc) at a specified coordinate is expressed as:

$$R = \sqrt{\sum_{i=1}^{N} R_i^2}$$

where Ri is the corresponding maximum response of the ith mode at the coordinate.

• CQC: complete quadratic combination. The estimated response is expressed as:

$$R = \sqrt{\sum_{i=1}^{N} \sum_{j=1}^{N} R_{i} \rho_{ij} R_{j}}$$

where ρ = the cross-modal damping coefficient.

Note:

* When some of the modes are closely spaced, the SRSS method may grossly underestimate or overestimate the maximum response. Large errors have been found in particular in space models in which the torsional effects are significant. The term "closely spaced" may be arbitrarily defined as the case where the difference between two natural frequencies is less than 10% of the smaller frequency.

* The CQC method is a more precise method of combining the maximum values of modal response.

* The two methods are identical for undamped models (x = 0).

The seismic analysis for this example is done according to the ASCE/SEI 7-16 Code; you may select other Codes, e.g., Eurocode 8, NBC-Canada, etc.

ASCE Standard (ASCE/SEI 7-16	5)	\times				
Select another code						
Minimum no. of modes to consider	: 5 of 10					
Earthquake direction : 🔀	1					
Site class [S]	B • 2					
Importance factor [le]	1.00 - 3	Cancel				
Response modification factor [R	1 1.5 4	OK				
Long-period transition period [TL]	1 5. 5					
Mapped spectral response acce	leration :					
At short periods [Ss]	0.26					
At a period of 1sec [S1]	0.26					
- Scaling of results						
C No scaling	Total seismic dead load [W]	630.98				
• Scaling (85%)	Period [T]	0.7253				

select 2 = Parameters in the side menu:

- 1. Specify the direction that the earthquake is applied. Select one of the global directions or define a vector as a combination of the three global directions. All mode shapes are used no matter in which direction the earthquake is applied. However, the modes which have deflections in the applied direction will dominate.
- 2. Specify the soil type as per Table 20.3.1.
- 3. Define the Importance factor as per Table 11.5.1.
- 4. Define the "Response modification coefficient" as per Table 12.2-1.
- 5. Specify a value of TL, calculated according to Section 11.4.5 and the Figures in Chapter 22.
- Specify the mapped maximum considered earthquake spectral response acceleration at short periods (Ss) and at 1s (S1) as detailed in Section 11.4 and the Maps in Section 20.
- 7. Referring to Section 12.9.4, when the base shear calculated from the modal shape analysis is different than the base shear calculated according to the Equivalent lateral force procedure of Section 12.8, all corresponding responses, including moments and forces are adjusted accordingly.
- to display the modal results, click <u>Display ta...</u> and select **Display modal results**



							RTI		TWARE DEVELOPMEN
1. ре	eriod (secon	ds)							
2. Pa su	rticipation m should b	factor: a e greater	factor re than 90	eflecting 1%.	the rela	tive influ	ience of	f the mode	shape.
3. su	m of extern	al forces	in all glo	obal dire	ctions.				
4. th	e root of th	e sum of	the squa	ares of th	ne horizo	ontal forc	ces.		
click	📕 Story da	ata to d	isplay th	ne story o	data resu	ults at ead	ch level:		
∘ Drift	Calculation	5							
Story d	lrift calculations								×
The	ASCE Standard (/	ASCE/SEI 7-16	5)						
Ampl	ification factor: 4		Ма	ax. drift/h : 🔽	.025		Height di	rection : 🔀 💌	J
No.	Level (m)	Height (m)	Drift (mm)	Max. Defl.	Min. Defl.	X1-Drift	X2-Drift	Weight(ton)	1
×	0.00	3.00	5.5	14	14	5.4	13	106.60	
V V	2 6.00	3.00	12.9	4.6	4.6	12.6	3.1	106.60	
\checkmark	3 9.00	3.00	17.3	8.9	8.9	16.7	4.4	106.60	
V	4 12.00	3.00	19.2	13.7	13.7	18.5	5.1	106.60	
	6 18.00	3.00	19.4	18.6	18.6	18.6	5.5	97.98	
		0.00		20.2	20.2		0.0	01.00	
			N						
	A check ma	rk that		The story o	drift				
	story drift in	at the		result	on				
	than the allo	wable							
∘ Stiffn	ess and ma	ss center							
== RIG	GIDITY AND MAS	s centers (U	nits: ton, me	ter)			_		
Exit C	Goto Print Co	ру		1	5 1 1 11		5.14		
No	. Level	<u>Ma</u> X1	ass	X2	Kigidity X1	×2	Differ DX1	DX2	
(D 0.00						Drif	DAL	
1		3.442	4.0)56 2	2.998	4.238	-0.444	0.182	
4	0.00	3.442	4.0	156 3	2.330	4.057	-0.444	0.041	
	o 3.00	J.442		100 .		4.034	0.441	0.030	
	4 12.00	3.442	4.0	156 2	2.996	4.093	-0.446	0.037	

o Story shear forces and moments

SHEAR FORCES/MOMENTS (Units: ton, meter) -								
Exit Goto Print Copy								
No.	Level	Story f	orces	Bases	shear	Story m	oments	
	-	F1	F2	V1	٧2	M2	M1	
0	0.00			55.78	16.69	740.57	230.61	
1	3.00	3.48	0.34	54.07	16.38	577.47	180.58	
2	6.00	7.68	1.11	49.55	15.32	420.18	131.50	
3	9.00	10.22	2.13	42.32	13.22	275.45	85.59	
4	12.00	12.20	3.31	32.22	9.94	150.66	45.94	
5	15.00	14.73	4.56	18.21	5.38	54.62	16.14	
6	18.00	18.21	5.38					

ATIR

ENGINEERING SOFTWARE DEVELOPMENT LTD

 \times

Stability coefficient (Theta)

Stability Coefficient (theta) The ASCE Standard (ASCE/SEI 7-16) Height direction : 🔀 🖃 Reduction factor : 0.25 No. Level (m) Height (m) Drift (mm) Weight(ton) Total Shear(ton) Theta 0.00 \checkmark 1 3.00 3.00 5.5 630.98 55.78 0.0052 V 2 54.07 0.0105 6.00 3.00 12.9 524.38 V 3 9.00 17.3 417.78 49.55 0.0121 3.00 🗸 4 12.00 3.00 19.2 311.18 42.32 0.0118 V 5 0.0103 15 00 3 00 19.4 204.58 32 22 \checkmark 6 18.00 3.00 18.7 97.98 18.21 0.0084 A check mark that The stability indicates that coefficient, Theta Theta is less than the allowable Week story \times Weak stories calculations The ASCE Standard (ASCE/SEI 7-16) 0.6 mPa Height direction : 🔀 🖃 Allowable shear stress (concrete) : Allowable shear stress (steel) : 150. mPa No. Level (m) Height (. X1-Shear Ratio X2-Shear Ratio 0.00 🗸 1 3.00 3.00 67.29 1.00 73.37 1.00 **√** 2 6.00 3.00 67.29 1.00 73.37 1.00 9.00 67.29 🗸 3 3.00 1.00 73.37 1.00 🗸 4 3.00 12.00 67.29 1.00 73.37 1.00 \checkmark 5 15.00 3.00 67.29 1.00 73.37 1.00 **√**6 18.00 3.00 67.29 73.37 Week story A check mark that ratio calculation indicates that the results week story ratio is less than the allowable

Seismic Analysis

ATIR ENGINEERING SOFTWARE DEVELOPMENT LTD o Soft story Soft stories \times The ASCE Standard (ASCE/SEI 7-16) Height direction : 🖂 🖃 0.7*Ku1 0.8*Ku123 Stiffness(K) Ratio Remark No. Level (m) Height (m) 0.00 V 1 3.00 3.00 131.0 36.99 32.56 3.54 √ 2 √ 3 6.00 3.00 52.8 26.68 25.41 1.98 9.00 3.00 38.1 21.82 20.87 1.75 **√**4 12.00 3.00 31.2 18.21 1.71 5 15.00 3.00 26.0 14.76 \checkmark 1.76 **√**6 18.00 3.00 21.1 Soft story ratio A check mark that indicates that the soft story ratio is results less than the allowable 🏹 Update re... in the side menu to Create static load cases from the modal results click and append them to the regular load cases: Create / Update STRAP static results files \times RSS over modes 1 • to 5 • Load name : RSS , DIRECTION:X1 Mode shapes Deactivate mode shapes (5 mode shapes deactivated) ΟK Cancel The program creates a fictitious load case representing an earthquake acting in the X1

direction, created by combining the mode shapes according to the code. The step should be repeated for the X2 direction, but the following steps assume that only this one case was created. In addition, we have not considered the minimum eccentricity required by the code.

•	display only the main model: select Display. Submodel instances and remove all
•	
•	click diawresult and select the options in the following menu:
	Graphic display
	Display type Wall results
	Result type Moment in x
	Load case
	C Combination 3 - RSS , DIRECTION:X1
	O Envelope
	Parameters Max result will be scaled as: 15 on
	Display only values greater than 0 % of max. result
	Display the result diagram in: C Screen plane 💿 Result plane
	 Hatch the result diagram Average results Results at: Design units Wall total
	Geometry lines type: 💿 Solid 🔿 Dashed
	OK Cancel
	The program displays Moment in the X direction for each wall design unit.



create combinations: select <u>Combinations</u> in the lower side menu, <u>III+1 Define/rev...</u>
 the upper side menu and create the following combinations:

No.	Title	1:Dead	2:Live	3:RSS ,DIRECTION:X1
1	ULS	1.4	1.6	
2	EQ_+X1	1	0.2	1
3	EQX1	1	0.2	-1

in