Seismic analysis of an ultra-thin concrete Shell

Guilherme Nunes Godinho

g.guilherme77@gmail.com

Abstract

Ultra-thin shells are a type of structures that consists in a bidimensional element with reduced thickness which is very efficient in transmitting loads through the section plane. This behaviour is called "membrane behaviour". Among one of their greatest advantages is the possibility to cover large spans that wouldn't be possible in other structural solutions such as slabs, without the introduction of intermediate supports or pre-stress.

In the last few decades, society has become more and more aware of the importance of seismic events. When it comes to earthquakes it's not about "if it will happen" but "when it will happen", so engineers have been developing technologies to improve and study the dynamic behaviour of structures when a seismic event comes to happen.

This thesis intends to evaluate the seismic behaviour of a pre-casted ultra-thin concrete shell. Firstly, the approach and modelling are made from a static perspective, where the assumptions made, regarding the model, are tested and verified. Secondly, a seismic, i.e., an acceleration, is applied to the structure to quantify and evaluate if the tensions and the displacements are withing the reasonable limits.

Keywords

Ultra-Thin Shell, Seismic analysis, Response Spectrum, Frequency, Vibration modes

1.Introduction

Nowadays, with an easier access to information, people have become more and more informed and conscious about the harmful effects an earthquake can cause on a community if some factors are not considered. It is why, and especially since the 60's of the last century, the governments and engineers started developing the seismic regulations that are used today. It's still not possible to predict in long term when exactly an earthquake is going to happen, so the best tool to avoid disastrous consequences is to guarantee the structures are well designed and constructed to withstand these actions so the maximum number of lives can be saved, and ideally with a minimum number of damages in the buildings. The objective of this dissertation is to evaluate the seismic behaviour of a pre-casted ultra-thin reinforced concrete shell using a pre-defined shape that is intended to be used as a roof to an urban playground

2. Earthquakes and Structures

As mentioned before earthquakes are a natural phenomenon consisting of an abrupt release of tension energy between the soil rocks that leads to a movement of the soil (acceleration), causing the foundations of the buildings to absorb those accelerations and transmitting them to the whole structure above. In this case of study, the area of interest is Lisbon, Portugal. Portugal is characterized to be a country with moderate seismic activity [1]. Its location is special as it is not only close to tectonic plates boundaries, but also its territory has some important faults that are active from a seismic perspective. This leads to the definition, by Eurocode 8, of two different kind of seismic actions that should be considered in Portugal. The type 1, or the "distant earthquake", is related to the inter-plate activity and in Portugal mainland this activity is mostly due to the Africa-Eurasia plate. The type 2, or the "near earthquake", is mostly because of geological faults and are smaller in magnitude than the type 1.

3. Shell Structures

As already stated, a shell structure is a bidimensional element with a reduced section (thickness) comparatively to the other dimensions, which allows the shell structures to cover large areas with multiple shapes and forms, without the necessity of having intermediate supports. Until 20^{th} century shells were mostly made of masonry, but with the emergence of the reinforced concrete, they began to be widely used assuming multiple purposes. This solution was also expensive, because the labour work required some expertise, and the formwork was hard to reutilize.

4.Modeling

The geometry of the shell used in this project was already defined as an elliptical paraboloid shape. It is supposed to be built with high resistance concrete, reinforced with special fibres.

4.1. Generalities

The shell was first drawn in AutoCAD. Then it was imported to the finite element method software SAP2000. Since SAP2000 does not allow to automatically generate the area elements in a pre-defined contour, they were put one by one, small enough so that the structure kept the curved shape.

Since it is an ultra-thin shell, the ratio radius-thickness should be around 1/200, so it was considered a section thickness of 5cm.

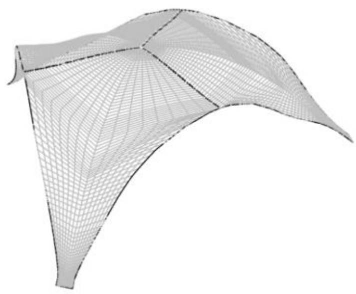

Figure 1. SAP Model

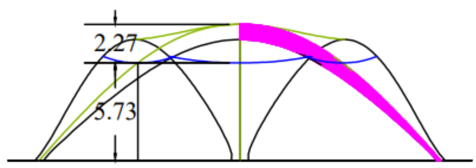

Figure 2. Frontal View

4.2. Special Joint

As referred previously, the shell is pre-casted. This has particular interest as it is not possible to build the shell entirely and then transport it to the local, where it is supposed to be installed, leading the shell to be built partially and then assembled in site. To model this, the structure was divided in 3 parts and their connections were modelled as being "gaps" joints.

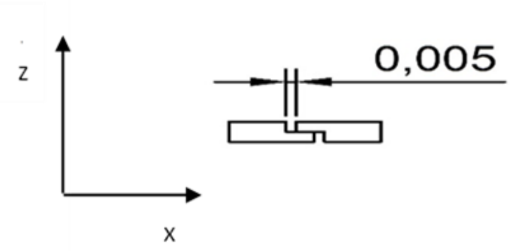

Figure 3. Inter-panel connection

In this joint all 3 translations, but one, are blocked. The translation through the X axis is permitted and the initial opening is 5mm. The stiffness of the joint was to be high enough, so that there weren't large movements, but a very high value could create numerical problems and high-tension peaks around the links. The value considered to this was 12000kN/m and it was verified how the vertical displacement at the top of the structure would be affected.

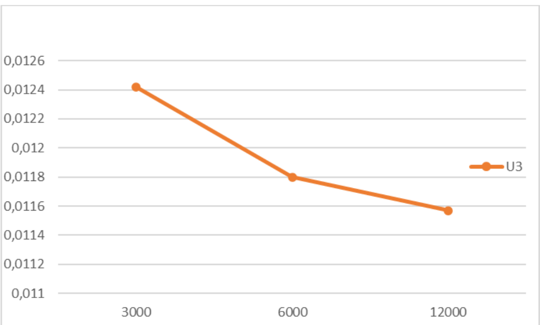

Figure 4. Stiffness vs Vertical displacement

4.3. Building Process

In this topic it was evaluate whether the building process has a major influence in stress levels or not. To do so 2 analyses were made: 1) Initially with the gap open and only the self-weight applied + gap closed and the remain loads applied.

2) Gap closed and the Psd applied entirely to the structure.

The static actions were the self-weight of reinforced concrete ($25kN/m^3$), permanent load of $1kN/m^2$ and a live load of $1kN/m^2$.

The combination of actions was the ultimate limit state of $1{,}35\ \times (G + R_{cp} + 1{,}5 \times S_c)$

The results made clear that the building process has no major influence and the peak values of stresses remained in the same magnitude order and in the same areas with the deviations being around mean 5%. The peak values were for the S22 compressions, as the figure 5 shows. It is important to refer that along the project some high values of tensions were obtained in the supports. Since the supports were not studied in this project that question was not taken into account.

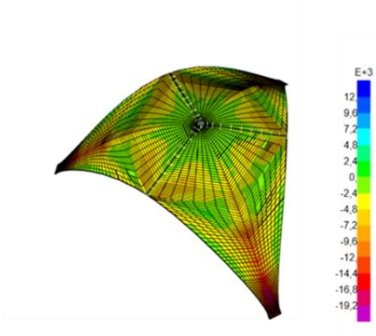

Figura 5. Max compressions S22 due to Psd

5.Seismic Analysis

This second part of the analysis refers to the seismic analysis of the structure.

5.1. Earthquake Action

To estimate the earthquake action linear response spectrums were used. For the definition of the response spectrum, some values are needed to be defined. The first was the behaviour factor. This factor depends mostly on ductility and since it was only used a linear response spectrum and a linear analysis this was considered to be 1. The shell is to be installed in Lisbon, so the soil was defined to be type B. According to regulation Lisbon is in the seismic zone of 1.3 and 2.3 for type 1 or 2 seismic actions respectively. This leads to accelerations of 1,5m/s^2 and 1,7m/s^2 respectively [2]

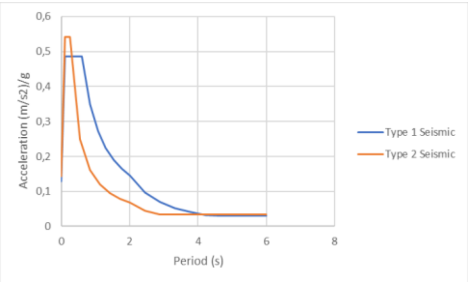

Figure 6. Response Spectrum

5.2. Frequencies and vibration modes

Running a modal analysis and obtaining the natural frequency of vibration, it is clear that the most conditioning seismic action is the type 1, based on the response spectrum graphic shown before, the first vibration frequency of 1,09Hz stays in the range <1 second of period where the acceleration of type 1 is higher.

In fact, the first and second vibration modes are similar since the structure has a symmetry in the X-Y plane and has the most participating mass of almost 50% of the mass vibrating in each direction for the two first modes.

For the X direction in the table 1, the 3 most important vibration modes regarding the mass participation factor are presented.

Table 1. Most relevant modes in X direction

Mode	f(Hz)	T(s)	Ux
1	1,09	0,92	0,49
53	33,33	0,03	0,09
59	33,3	0,03	0,06

The first mode corresponds to the natural frequency in which approximately 50% of the mass is vibrating in the X direction.

The 53^{th} mode is the mode in which the rotation in the Y direction has the highest value.

For the Y direction the situation is equivalent with the 3 most important modes being similar to the ones in the other direction as shown in the table 2

Tab*le* 2. Most relevant modes in Y direction

Mode	f(Hz)	T(s)	Uy
2	1,09	0,92	0,50
54	33,33	0,03	0,09
58	33,3	0,03	0,06

The 3 most important modes in Y are identical to the ones in X, as the 2^{nd} mode is the same as the 1^{st}, which concludes that the modes in Y are similar to the X in pairs.

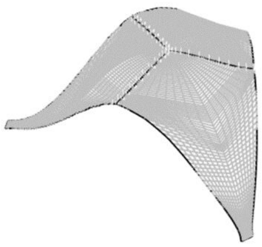

Figure 7. 1st Vibration mode

5.3. Tensions and Displacements

In this last part of the thesis the seismic effect is evaluated from the tensions and displacements perspective. Note that because of the similarity of the structure in the X and Y directions, the changes of the direction of seismic, will only change the areas where the tensions and displacements are higher, not the magnitude of the value itself.

5.3.1. Base Reactions

One quick way to confirm if the type 1 seismic is more important than the type 2 is the base reactions. In fact, as the table 3 presents, the base reactions for the type 2 are nearly 50% of the ones for the type 1, so the theory is confirmed.

Table 3. Base Reactions

Seismic Type	Fx (kN)	Fy (kN)
Type 1	56,00	56,49
Type 2	29,01	29,10

5.3.2. Tension Analysis

In this topic it is shown the effects of the seismic in the structure from the tension's perspective.

Being the seismic action a soil acceleration that is transmitted to the structure, and according to Newton's 2nd Law, stating that the force is proportional to the mass and acceleration, in this case of the shell structure the reduced thickness leads to a relatively short mass value of approximately 38tons, so it is expected that the seismic doesn't cause a high increase in the tensions.

In fact, and according to the SAP analysis it is possible to divide the structure according to two situations regarding the tensions. Notice that the modal + seismic analysis will only give absolute values for the tensions.

The S11 tensions will have their peak values mostly in two areas of the shell: in the upper parts of the supports and in the inter-panel connections. The S22 will have their highest tension values in the external supports. A test was made showing that if the external supports are free to the X and Y translation, the tensions will go down.

5.3.3. Load Combinations

The load combination for seismic action, according to the EC1 is the following:

$$P = G + R_{cp} \pm S + \Psi_2 \times S_c$$

Being G, Rcp and Sc the static loads, as referred before; S the seismic action and the $\Psi_2 = 0$ since it is a non-accessible roof.

In this chapter both directions, X and Y, are studied separately, so that it can be possible to understand more accurately the effects.

The analysis shows that from an overall perspective the shell will have average tensile tractions within the 5-7MPa range. Besides that, the inter-panel connections and the upper part of the supports are the other areas where the tensile tensions could go higher specially in the S22 component.

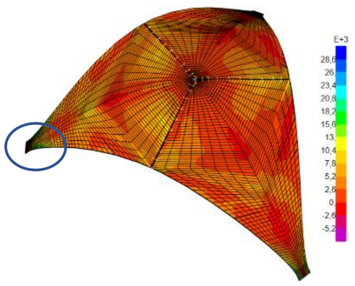

Figure 8. S11 max tensile tensions

As the picture shows the highest tensile values will occur in the lateral parts of the external supports. The highlighted one is orientated into the X direction.

For the compressions the S11 will essentially be higher in the upper part of the supports and the S22 tensions will have peak values in the external supports. An expedite comparison with the maximum values obtained in the chapter 4 concludes that the seismic action will increase the values for the higher compressions. Since the concrete is a high resistance one, this situation is within the limits.

In the Y direction the situation is identical to the X direction, but the load distribution through the 3 supports is more balanced.

Once again, the biggest tensile tensions will appear at the supports whereas in the shell they are in the same range as the previous topic.

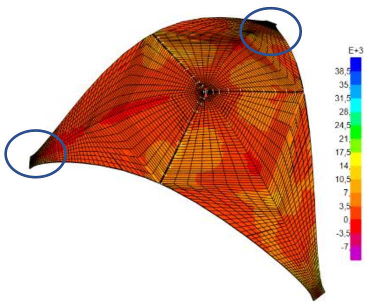

Figure 9. S22 max tensile tensions

On the compression side the higher values will be at the top of the two supports not oriented in the X direction, and at the external supports.

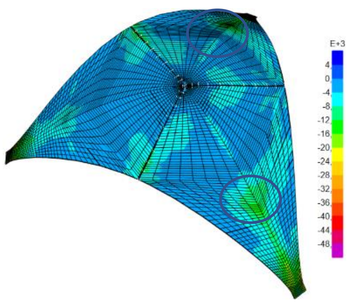

Figure 10. S22 max compression tensions

Again, comparing the peak tensions obtained in the chapter 4 with the results in this analysis, the seismic will increase the compressions.

5.3.4. Displacements

To conclude the analysis, it was studied the vertical displacement through 2 axes and according to the axis selected the seismic was in X direction or Y direction.

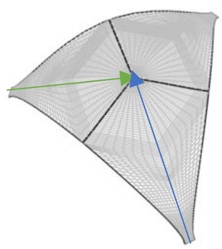

Figure 11. Vertical Displacement along two axes

As the pictures presents, the axis green is studied with an X direction earthquake. The blue axis is with Y direction.

The graphic below shows the displacements are along both axes, showing that in the middle sector there is a signal inversion.

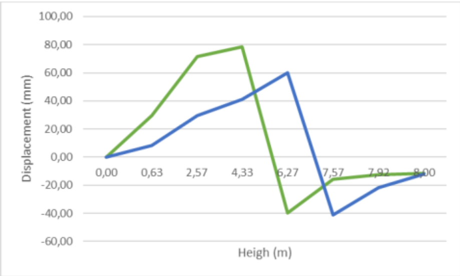

Figure 12. Graphic of vertical displacement

6.Conclusion

To conclude this thesis made clear that this structure is statically and dynamically stable. The seismic action doesn't have a significantly high amplification of the tensions and besides the external support situation the values are within the limits for tensile and compressions of the material. To solve this support peak of tensions a good solution could be to gradually increase the shell thickness from the middle sector until the base. Also, the peaks and the inter-panel connections could be solved if more nodes as links are added to the model since in the reality the whole edges will be connected.

7.References

[1] *"O que é um sismo."* (n.d.). IPMA. https://www.ipma.pt/pt/educativa/faq/geofisica/sismologia/faqdetail.html?f=/pt/educativa/faq/geofisica/sismologia/faq_0001.html

[2] EN 1998-1 - Norma Portuguesa - Eurocódigo 8: Projecto de estruturas para resistência aos sismos, 2009.