Interactive Structural Analysis and Form finding

Odysseas Georgiou
Structural Engineer

University of Bath
odysseas.georgiou@hub.com.cy

Summary

This paper re-approaches structural engineering through an interactive perspective by introducing a series of tools that concatenate parametric design with structural analysis, thus achieving interoperability between the architectural shape and its structural performance. Furthermore, this paper demonstrates how the design can be realised into an efficient structural form by applying novel techniques of form finding through the exploitation of the created tools by iterating between ranges of structurally optimum solutions.

Keywords
Structural engineering, parametric design, interoperability, free form, form finding, analysis, interactive, performance based design

1.   Introduction

The rapid advance in CAD technology has enabled architects to overcome the traditional design boundaries and to transform any imagined shape into a persuasive building. In this context, structural design is lagging behind and engineering's engagement with architecture is still restricted. This traditional approach cannot keep up with the modern design process and the engineer is unable to give feedback to the architect's design, often stalling the design process. While a large variety of tools serving architectural geometry, such as parametric modelling, is available for use by architects, allowing limitless capabilities and speed in design, the engineering industry remains adherent to traditional structural analysis and design techniques.

This paper introduces a novel design procedure through a series of tools that interactively manage and form-find structures. An overview of combining a parametric design software, Grasshopper 3D [1] with a structural analysis software, Autodesk Robot Structural Analysis [2], through the use of computer programming is presented. This combination enables the engineer to retain better control over his designs by employing a performance based approach and it speeds up the design process while allowing for the exploration of new optimum structural solutions [3, 4]. By extending the capabilities of parametric design to include and implement structural analysis, the engineer can move away from the traditional ways of structural thinking and relax the technical boundaries. The results deriving from a structural analysis need no longer be single solutions to problems but parameters that feed into the architectural form and conclude to an optimum shape.

Parametric modelling has evolved in the last decades through the application of advanced computing to architecture and it refers to designing an object by describing the relationship between its parameters. This allows the re-definition of the final output by manipulating the components of the object. An object is described by its geometrical constraints or by its degrees of freedom as well as by its dimensions and the equations that define their correlation [5]. Such extension, serving geometry and consequently architectural design, is not however available, in a form relevant to the work of a structural engineer. To demonstrate the capabilities of a structural extension to parametric

design, two examples of distinct design cases are presented in this paper. Both cases employ the performance based approach by first interactively visualising structural analysis results and then utilising them to iterate between a range of structurally optimum solutions which respond to the designer's control.

2.   Parametric Truss form finding

To illustrate the means of performance based structural design a case of a two dimensional problem is considered first. This example utilizes a framework that generates a plane truss geometry following the interactive user input. The framework then analyses the geometry in a structural analysis software, returns the output in conjunction with the initial geometry and optimises the shape following the user objectives. For this purpose, McNeal's Grasshopper 3D and Autodesk's Robot Structural Analysis, were linked together using the C# programming language.

2.1   Interactive Framework

Grasshopper 3D is a plug-in built in .NET framework to access McNeal's core software, Rhinoceros 3D (CAD) [6] in order to control and manipulate geometry in a generative manner. The functionality of Grasshopper 3D (GH) can be extended by writing code in C# or VB .NET programming language to create custom components. In parallel, Robot Structural Analysis (RSA) allows the interaction with other software and the use of its Calculation Engine through its Application Programming Interface (API) [7]. General CAD software that handle mainly geometry and FEM software used by structural analysts, understand and control input geometry in a different manner. What makes a significant difference in the representation of a structural model is the need to attach structural attributes to each of its elements, while the geometric model can be purely described by its topology. In addition, the numbering of each element and its global orientation in relation to its local axis are crucial points in the definition of a structural model, whereas this might not affect the generation of the former.

The truss geometry was first defined parametrically so that all of its attributes could be controlled in an interactive manner. The geometric attributes included the truss' span, depth, the number of bays and the truss type (Brown truss, Pratt truss and Vierendeel truss). Structural constraints such as supports, bar sections, material properties and loads needed to be included in the model. This attributes were also controlled in an interactive manner, meaning that the user is able change them using control sliders or can design and manipulate loading curves (Figure 1). This was done by using RSA's relevant interfaces and members in a similar way, as the user would define them using the software's Graphical User Interface (GUI). When a structural model included all the information needed to be analysed, the calculation engine interface was called and the desired results were generated and returned inside the GH custom component.
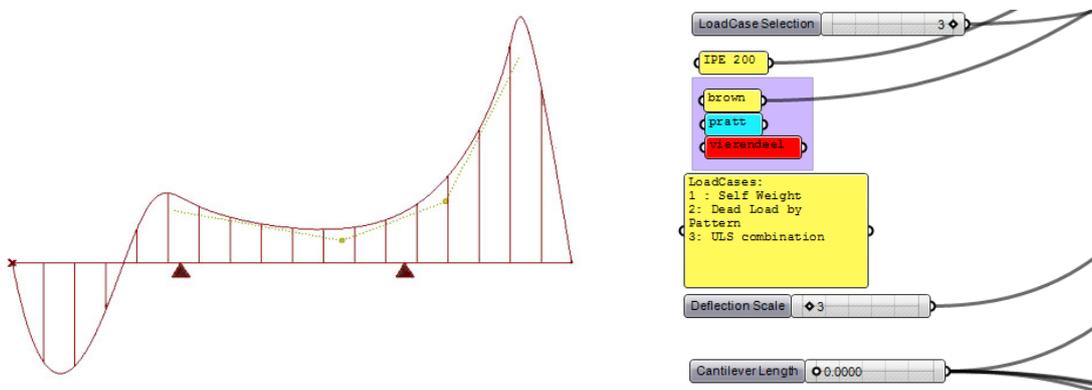

Fig. 1 - Defining the truss' parameters in Grasshopper's sketchpad.

The results were returned by utilizing the appropriate interface, depending on the type of the results and the reference to the structural entity. In the current case, nodal deflection and bar force data were returned from the analysis. These results could be used as parametric outputs to enable structural analysis visualisation in Rhinoceros 3D environment (see Figures 2 and 3).
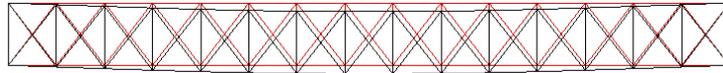

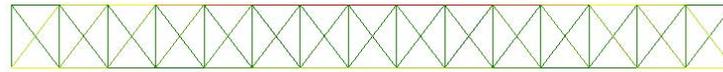*Fig. 2- Visualisation of deflection under self-weight in Rhino3D*

Fig. 3- Visualisation of the distribution of forces in each element under self-weight in Rhino3D

2.2   Parametric form finding

In a further step, the generated results were used to form an optimised geometry that could respond to the user's input parameters. A method leading to a more efficient form of a truss is to adjust its shape to its stress conditions. Trusses are used to bridge spans and therefore the most critical form of force that they are subject to is bending moment. Bending moment introduces axial forces on to the chords of the truss and consequently members are more stressed in areas of high bending moments. The stress induced in a member of constant section under pure bending force equals to:

$$\sigma_{xx} = \frac{M_{yy}}{I_{yy}} z \quad (1)$$

where σ_{xx} is the normal stress, M_{yy} is the applied moment, I_{yy} is the section's moment of inertia relative to the bending axis and z is the distance from the members neutral axis to the point that the stress is calculated.

The algorithm applied to the GH component adjusted the truss's depth according the initial stress conditions, caused by the influence of bending moment, until the sections of the truss chords were utilized to the larger extent. This lead to a form of truss associated with the shape of the bending moment diagram (see Figure 4).
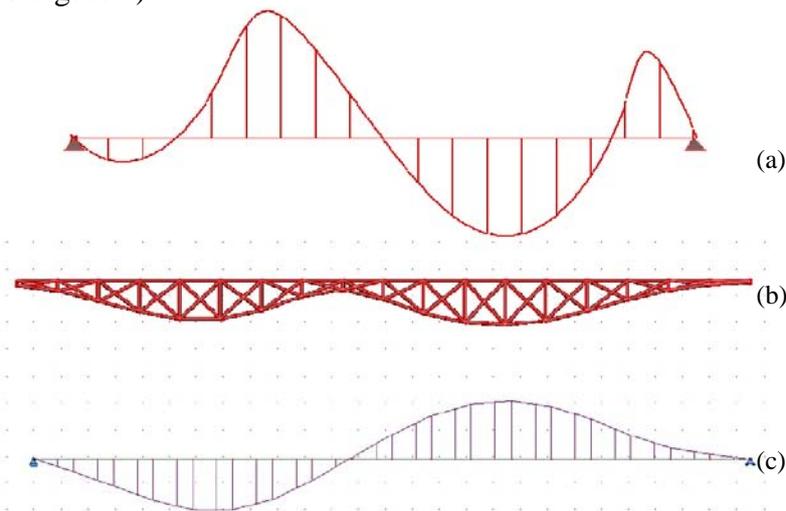

Fig. 4 - Resulting output of the algorithm (b) in relation to the bending moment diagram (c) under an arbitrary load case (a)

The designer could alter all the parameters that were defined previously to form the truss and the component will shape the truss according to those parameters. This could offer additional control to the engineer while moving away from the distinct optimum solutions to the problem (see Figure 5).


(a)
(b)
(c)
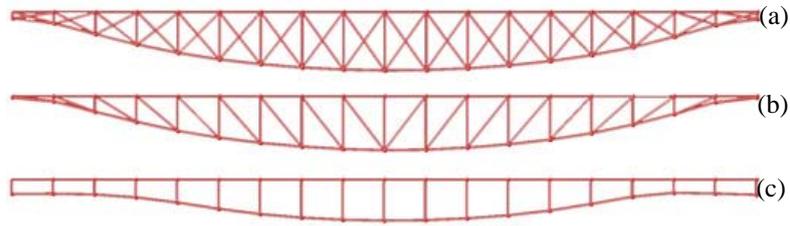
Fig. 5 - Resulting optimised shapes for Brown truss (a), Pratt truss (b) and Vierendeel Truss (c)

3.   Interactive grid form finding

A more complex case is investigated in this section, which relates to a free form surface. In a similar manner as in the two-dimensional problem above, the free form surface's geometric representation was linked to the structural analysis software through a framework that was able to visually present results as well as to further exploit them for the generation of efficient forms.

3.1   Free form surface analysis

A free form surface was first defined in the form of NURBS [8], which is the most suitable method for the digital representation of such surfaces. Rhinoceros 3D and consequently Grasshopper 3D are based on NURBS modelling. Although the only way to structurally analyse a surface is through the use of discritized or finite elements (planar triangles or quads) it was found practical to first represent it as continuous (i.e. as NURBS) in order to freely control it in three-dimensional space (Figure 6).
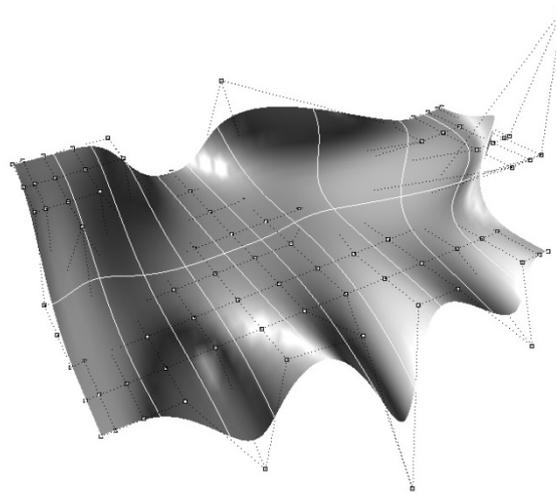
Fig. 6 - NURBS surface representation in Rhino3D

A finite element mesh is defined by quadrilateral or triangular Faces (flat facets) bounded by edges which in turn are defined by end vertices touching the original surface. The mesh is therefore an approximation of the continuous surface. For the surface's conversion to a mesh, a standard GH component was used. GH's mesh output is a connectivity matrix of vertices, edges and faces. This information needs to be decomposed in elements that can be read by the structural analysis component. The mesh vertices were translated in a list of points in 3D space, which were then

translated in structural nodes inside the analysis component. The connectivity of each face was then used to create arrays of nodes by selecting the sets that comprise each face out of the list. Finite elements could then be created by utilizing the arrays created using appropriate methods in RSA's finite element interface. Structural properties needed to then be applied, which were treated in a way that allowed them to be controlled parametrically by the user through the GH graphical interface. The properties relative to FE and essential to the analytical process are the type of the elements, the material and the elements' thickness. For the current example, 3-node planar elements with constant thickness of 300mm and C30 concrete material properties were used. The model's supports were initialised to the edges by selecting all the nodes that lie on the mesh's outline. The border of the mesh was isolated by utilizing the topological information of the mesh. A routine was formed to run through all the edges that form the mesh's topology and select the ones that were only connected to one face. The vertices that belong to those edges were output to the analysis component and are selected as nodes that contain support information. The topological information is lost when the mesh is translated to FE for the structural modelling and therefore the index of each support vertex needed to be found from the list of nodes that were already drawn in the model. Objects needed to be selected individually for attributes to be assigned on them.

After the structural information database was complete, the calculation interface could be called and results could be output for each finite element. It was important at this stage, that the structural coordinate system was aligned with the environment in which the results are to be visualized. For that reason, the user was able to select the direction that the FE results are aligned to. For the purpose of graphical representation of the FEA results, each face was translated into a triangulated surface while keeping the topological information of the mesh. Consequently, the analysis results were mapped with each surface facet and their values could be tagged accordingly or coloured using a *GH Gradient* component. This resulted to the interactive representation of the impact of change on the geometry or the constraints of the initial NURBS surface in Rhino's Viewport (see Figure 7).
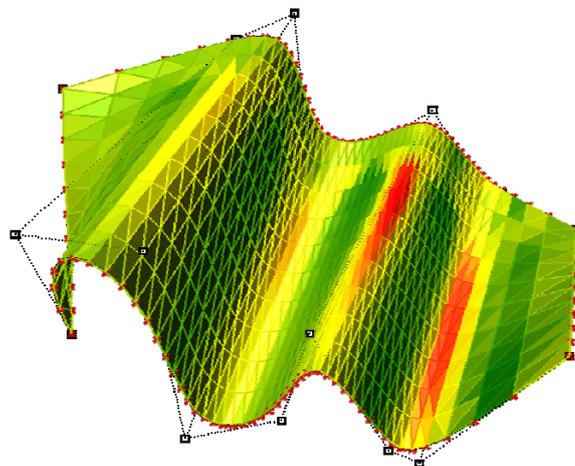

Fig. 7 - Graphical representation of surface stress ratios in Grasshopper 3D

3.2   Grid optimisation

In the context of interactive optimisation, an interactive technique for designing grids is introduced. This is based on the directions of principal stresses that occur in a continuum shell. Principal stresses are the components of the stress tensor that occur at each point of a continuum, which are purely axial, consequently their shear component equals to zero. The directions at which these stresses occur are called Principal Stress Vectors. These components share the maximum and minimum stress values and ideally, if a grid is aligned along their directions, it can replace the continuum [9, 10]. The influence of the stresses along those directions is evident in the case of a

simply supported beam. Figure 8a shows the tension stress trajectories and the cracks that develop in the concrete beam (perpendicular to the former) caused by the material's inability to withstand tension.
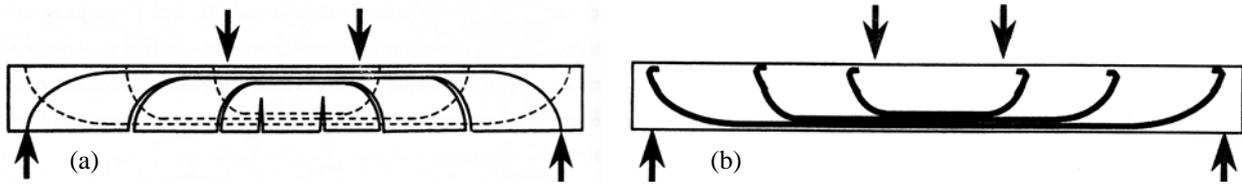

Fig. 8a - Cracks developed in a simply supported concrete beam (image: Kotsovos, M.D.) Fig. 8b-
Theoretical reinforcement arrangement following the tension principal directions (image: Kotsovos, M.D.)

The most efficient way to balance this weakness would be to incorporate steel rods along the tension trajectories developed by the external loading (see Figure 8b). However, because such arrangement is difficult to construct the usual reinforcement of a concrete beam consists of straight bars [11].

The first part of this procedure involved the use of the framework previously established to output values to be used in the plotting algorithm. In the current context, plane stress conditions were considered, i.e. cases where one dimension of the continuum is much smaller than the other two are only explored and thus the stresses that act in the through-thickness direction are neglected.
A linear elastic analysis was then carried out for stress values to be determined. The stress data was then extracted for multiple layers through the shell's thickness so that bending and membrane stresses were both taken into consideration. The principal stress data (principal stress magnitudes and vector directions) were sent back to GH to be used as data for the plotting algorithm (Figure 11).

A series of algorithmic routines were developed to plot the principal stress trajectories on a free form surface following the analysis output: The stress data had first needed to be mapped on each of the planar surface generated in GH. Starting from an arbitrary face in the mesh group, curves were drawn following each of the direction of the principal stress vector field. When a curve (which becomes a trajectory after the initial step) met a face edge, an intersection occurred which was crucial in determining the next step and the preferred face at which the plotting would restart. A precise intersection routine was formed in order to facilitate this.

Because the finite element analysis only returned one pair of stress results per triangle, there are sharp changes in direction at the triangle edges between adjacent faces. For that issue to be addressed, smoothing algorithms were developed that were able to choose the best possible principal route taking in account the set of principal vectors in four directions (including the exact opposite values of stresses) and the previously plotted trajectory. (Figure 9)
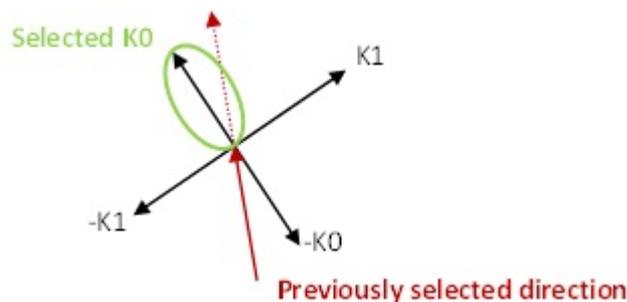

Fig. 9 - Direction selection algorithm

The plotting algorithm would go through all the available mesh faces that were holding principal stress data, by also avoiding the ones that had already been initialised, that is having a pair of principal stress trajectories passing through them. The finalised algorithm produced the grid shown

in figures 10 (a) and (b) below. The colour represents the stress distribution along the principal trajectories.
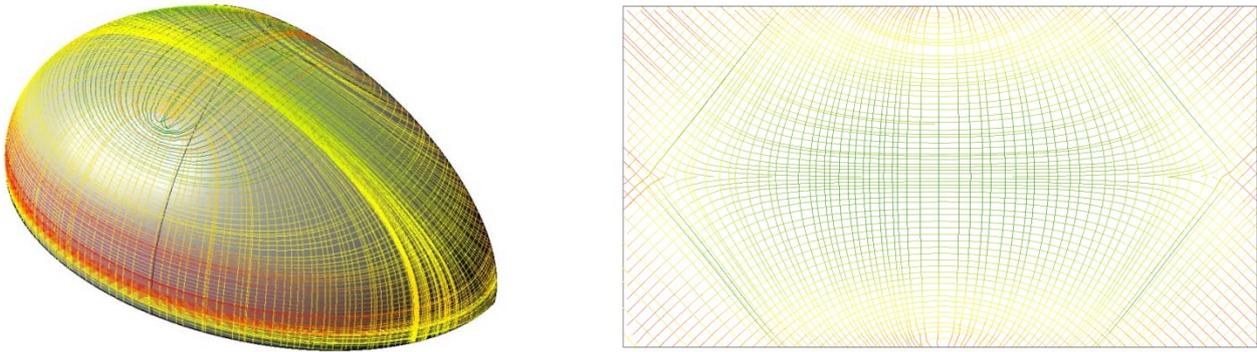
Fig. 10 - GH output principal stress grid (a) free form surface, (b) edge supported plate

The principal stress trajectories ouput was further post procesed by developing algorithms to produced a more evenly spaced plot by controlling the distance between each curve, merging the ones that were converging and removing the ones that were concurring. The post processing output is shown in Figure 12.
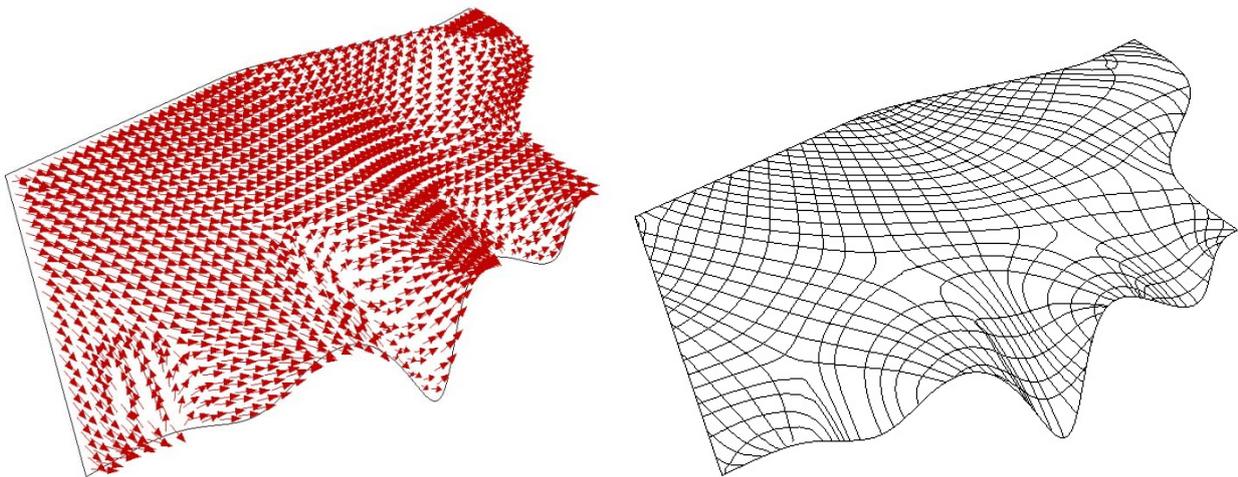
Fig. 11 - Maximum Principal Stress Vectors output in Grasshopper - Fig. 12 - Evenly spaced grid pattern on the test surface

4.   Conclusions

This paper presented the generation and application of design tools aiming to achieve a synergy between form and structure while at the same time leading to the creation of efficient structures for complex architectural shapes. This was accomplished by employing the power of parametric design combined with structural engineering software, liking them by computer programming. This combination enabled an interactive approach to structural design, a function which currently is sparingly applied for solving engineering problems. The application of the method was demonstrated through two examples: a simply supported two-dimensional truss and a free form surface. Both cases were interactively analysed responding to the designers controls having their form adapting to multiple variations of external conditions, as supporting conditions loading and material. The presented studies prove possible that optimum structural solutions need not be singular allowing the designers to iterate between solutions that respond to their performance

criteria. Furthermore, this creates the challenge for future adaptation of systems that enable integration of multi-diciplinary design teams to allow interaction of their dedicated parameters for the generation of multi-ojective optimum solutions.

References

1. Grasshopper- Generative modelling for Rhino [online]. *Available from: http://www.grasshopper3D.com/* [Accessed 25 June 2010].

2. Autodesk- Autodesk Robot Structural Analysis Professional [online]. *Available from:http://usa.autodesk.com/adsk/servlet/pc/index?siteID=123112&id=11818169* [Accessed on 20 June 2010].

3. Kristina Shea, Robert Aish & Marina Gourtovaia. (2005).Towards integrated performance-driven generative design tools. *Automation in Construction*. March 2005, 14(2).pp. 253-264.

4. Shepherd, P., Hudson, R., Parametric Definition of the Lansdowne Road Stadium. *International Association of Shell and Spatial Structures*. 2007, Venice, Italy.

5. Aish, R., Woodbury, R., Multi-Level Interaction in Parametric Design. *Bentley Systems Incorporated.*

6. Modelling tools for designers [online]. *Available from: http://www.rhino3D.com/cad.htm* [Accessed 01 July 2010].

7. Wikipedia, Application Programming interface. *In: Wikipedia: the free encyclopaedia [online]. St Petersburg, Florida: Wikimedia Foundation. Available from: http://en.wikipedia.org/wiki/Application_programming_interfac*e, [Accessed 2/07/2010].

8. Rhinoceros- What is NURBS? [online]. *Available from: http://www.rhino3D.com/nurbs.htm* [Accessed 01 July 2010].

9. Winslow, P., 2009.Synthesis and Optimisation of Free-Form Grid Structures. *Thesis (PhD). University of Cambridge.*

10. Michalatos, P., and Kaijima, S. Design in a non homogeneous and anisotropic space. *Symposium of the International Association of Shell and Spatial Structures*, Venice, Italy, December 3-6 (2007).

11. Kotsovos,M.D.,2005. Reinforced Concrete. Athens: *NTUA publications*.