Nonlinear Finite Element Analysis of Space Truss

Dr. Ahmed Farhan Kadhum
Building and Construction Engineering Department
University of Technology
E-mail: ahmedms1976@yahoo.co.uk

ABSTRACT.
This paper presents an analytical investigation which includes the use of three dimensional nonlinear finite elements to model the performance of the space trusses by using (ANSYS 11.0) computer program. The numerical results show very good agreement (100%) with experimental results, while the graphical option reflects the behavior of the structure under the applied loads because of the ability of this option to simulate the real behavior of the structure under these loads. Also finite element models of the space truss simulate the lateral deflection of the top chord members especially at the corners, and the twisting of the bottom chords.

Keywords: Space Truss, Top Chord, Bottom Chord, Finite Element, ANSYS 11.0, Shell Element.

1. INTRODUCTION.

Space trusses are popular in covering large open areas with few or no internal supports. Among their advantages are mass production, easy transportation, fast assembly, light weight and pleasant appearance. The last four decades have been a widespread use of space trusses and the development of many new systems, each with different features to attract users and gain a larger share in large-span structures markets [1].

Space truss systems can generally be put into two main groups:

1) Systems with **short chord** members joined together by node connectors; most space trusses available today belong to this group of systems. They generally consist of similar members of short length connected at the joints with similar nodes (that are usually sophisticated and expensive); and
2) Systems with **continuous chord** members that do not need nodes for their assembly;

in order to overcome the high cost normally associated with space trusses, system have been developed with jointing methods that do not rely on special node connectors. In order to achieve that the chord members of this group of trusses are made continuous across the joints and usually are connected directly together, either by bolting or welding.

Conclusions concerning optimum design of double-layer space grids were submitted by Agerskov H., 1986 [2] made from the results obtained in the investigation. The member density must be small. In addition to giving a small material consumption, this leads to a grid with relatively few nodal points and thus least possible production costs for nodes, erection expenses etc.

The results presented by Murtha-Smith E., 1988 [3] of the analysis performed on a hypothetical space truss and on a constructed space truss showed that progressive collapse could occur following the loss of one of several potentially critical members when the structures were subjected to full service loading. However, when the structures were evaluated using the American National Standard ANSI A58.1-1982, the structures were found to survive with a small margin safety. It was suggested that to improve resistance to progressive collapse the compression members and additional members along and adjacent to the column line should be design with higher factors of safety than those currently used, particularly in the middle half of the span.

A methodology was developed by Christopher et al., 1989 [4] to perform nonlinear postbuckling analysis of steel space truss systems. Structural behavior was modeled at the element level through appropriate stress-strain relationships. Nonlinearities due to member buckling or yielding were modeled using a tangent modulus in the evaluation of the element stiffness matrix. Results were obtained and compared for linear and nonlinear material behavior along with second-order geometric nonlinearity. The general results provide information on the failure mechanisms most critical to a particular truss system, and techniques which can be used to reduce the effects of the failure mechanisms were described.

Another new space truss system, named **Catrus**, had been developed at the University of Dundee by Ahmed El-Sheikh, 1996 [1]. The target was to design a system that; combines low cost with reliable structural behavior and provide solutions for many practical applications. The main feature of Catrus was: the top and bottom chord members were continuous across the joints, the members were directly bolted together without any node connectors, and chord and diagonal members were stacked above each other.

Another paper presented by Ahmed El-Sheikh, and H. El-Bakry, 1996 [5], a space truss system developed with the main objective of achieving a larger reduction in the overall cost of space truss without compromise in the structural reliability or the common case of construction. To achieve this goal, the (new truss) had a simple joining system that required no expensive node connectors, and a simple member splicing system that had almost no adverse effect on the truss's performance. The paper included an introduction to the system's feature and an experimental assessment of the claim that the new joining system of the truss did not entail any compromise in its structural efficiency or reliability.

A paper presented about the space truss by Ömer Kelesoglu and Mehmet ükler [6] with a general algorithm for nonlinear space truss system optimization with fuzzy constraints and fuzzy parameters. The analysis of the space truss system was performed with the ANSYS program. The algorithm multiobjective fuzzy technique was formed with ANSYS parametric dimensional language. In the formulation of the design problem, weight and minimum displacement were considered the objective functions. Three design examples were presented to demonstrate the application of the algorithm.

Jin-Woo Kim et.al, 2008 [7] presented a paper discussing the behavior of cable-tensioned and shaped hyper space truss, with consideration of the influence of removing some web members in two directions. Hyper shaped space truss was cable-tensioned at the strand of bottom chords with one diagonal on the ground; the essential behavior characteristic of shape formation was discussed by using a small-scale test model. Results of experiments and nonlinear finite-element analysis indicated that a planar, rectangular-arranged structure can be deformed to a predicted hyper shape, by the proposed cable-tensioning method. Also the feasibility of the proposed method for furnishing of a hyper shaped space truss had been presented, under the conditions of removed web members with both active diagonal and passive diagonal.

2. PARAMETRIC STUDY.

Verification is done in order to check the validity and accuracy of the finite element procedure. Thus, two space trusses (A and B) with available experimental results [1] have been analyzed here and the analytical results are compared.

The space truss Catrus is consisted from:

- **Top chord members:** rectangular hollow sections (RHS) are used in the top chord members of Catrus to resist their dominantly compression forces.
- **Bottom chord members:** the bottom chord members of Catrus are flat strips prepared. The use of flat strips provides a number of advantages including simple fabrication, simple splicing, and low tendency for joint rotation.

- **Diagonal members:** the diagonal members of Catrus are made of circular hollow section (CHS) tubes with their ends squashed as shown in **Fig.(1)**.

The space trusses of the experimental work are:

1. **Truss A** with overdesigned top chord members (by 30%). This truss modeled a practical situation in which the top members were moderately overdesigned to bring in some ductile behavior.
2. **Truss B** with underdesigned top chord members (by 40%). This truss modeled an impractical case and was only included to study the efficiency of Catrus trusses in accommodating cases of individual member buckling.

3. FINITE ELEMENT.

The space trusses A and B were analyzed with (ANSYS software computer program release 11.0) idealized by subdividing the structure into a number of elements as shown in **Fig.(2)**.

1. Top and Bottom Chord Members were modeled by **Shell-143** element which is well suited to model nonlinear , flat or wrapped, thin to moderately–thick shell structures. The element has six degrees of freedom at each node: translations in the nodal x, y, and z-directions and rotations about the nodal x, y, and z-axis. The deformation shapes are linear in both in-plane directions. For the out-of-plane motion, it uses a mixed interpolation of tensional components. The element has plasticity, creep, stress stiffening, large deflection, and small strain capacities. The geometry, node locations, and the coordinate system for this element are shown in **Fig.(3)**.

The element is defined by four nodes, four thicknesses, and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions [8].

2. Diagonal Members were modeled by **Link-180** is a spar that can be used in a variety of engineering applications. This element can be used to model truss, sagging cables, links, springs, etc. This 3-D spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translation in the nodal x,y, and z directions. As a pin-jointed structure, no bending of the element is considered. Plasticity, creep, rotation, large deflection, and large strain capabilities. Elasticity, isotropic hardening plasticity, kinematic hardening plasticity and creep are supported. The geometry, node locations, and the coordinate system for this element are shown in **Fig.(4)**. Also link-180 allows a change in cross-sectional area as a function of axial elongation. By default, the cross-sectional area changes such that the volume of the element is preserved, even after deformation. The default is suitable for elastoplastic applications.

Fig.(5) shows the idealization of the space truss with **shell-143** elements to model the Square Hollow Sections (SHS), bottom flat members, and **Link-180** to model the Circle Hollow Sections (CHS).

4. MATERIAL MODELING.

The top and bottom chord members, in addition to the diagonal members are steel sections and are modeled as elastic-plastic material. The steel material by ANSYS is modeled by two parts; linear elastic material model and the required values are:

- Elastic modulus (E_s)
- Poisson's ratio (v)

while the second part is bilinear inelastic to represent the stress-strain behavior of material as shown in **Fig.(6)** , and the input data which are needed for ANSYS are:

- Yield stress (f_y)
- Tangent modulus (E_{tan})

For the *Space Truss Modeling* Shell type element 143 is used to model the steel sections of the space truss; the material can have orthotropic properties corresponding to the element coordinate directions [8].

The input data of material properties in the ANSYS program are the modulus of elasticity E_x, E_y, and E_z (one value required), Poisson's ratios v_{xy}, v_{yz}, and v_{xz} (one value required), and the shear modulus G_{xy}, G_{yz}, and G_{xz} (one value required),

In addition, the yield stress (f_y) and the tangent modulus (E_{tan}) are required for the bilinear inelastic stress-strain behavior as in the reinforcing steel (rebars).

5. FINITE ELEMENT RESULTS AND DISCUSSION.

The present section highlights on the nonlinear behavior of space truss using (ANSYS program). ANSYS is a program intended for solving practical engineering problems. The idealization of the space truss is done by subdividing the structure into a number of elements as shown in **Fig.(7)**.

The word loads in ANSYS terminology includes boundary conditions and externally or internally applied forcing functions, for example: loads, displacements U_x, U_y, and U_z (DOF constraints), forces, pressures.

For the tested truss in this study, the displacements (DOF constraints) U_x and $U_y = 0$ to represent the hinge end, while the other end is a roller so just $U_y = 0$.

From the load-deflection curves of the two space trusses A and B analyzed by finite element ANSYS program, plotted by GRAPHER 1.09 software as shown in **Figs.(8 and 9)** and compared with the experimental results by Ahmed El-Sheikh [1]. Very good agreement can be seen between the experimental and analytical values as summarized in **Table(1)**, while there is a difference between the experimental and theoretical curves path and this is obvious in **Fig.(9)** and the reason for this manner that the analysis modeling used in this study could not catch the part of steel material behavior which is known as "necking" phenomena, which can be defined as: (after a critical value σ_y of the stress has been reached, the specimen undergoes a large deformation with a relatively small increase in the applied load. This deformation is caused by slippage of the material along oblique surfaces. After a certain maximum value of the load has been reached, the diameter of a portion of the specimen begins to decrease, because of local instability) [9]. In addition, after this certain maximum load which is known as UTS (Ultimate Tensile Strength) (denoted σ_t in these modulus). (Beyond that point, the material appears to strain soften, so that each increment of additional strain requires a smaller stress) [10] as shown in **Fig.(9)**.

The most critical part of curve obtained from analytical program which faces difficulty for idealization with the experimental is the strain hardening."It has been observed that even with the same method of measurement, there is still considerable scatter in the value of E_{st} " [11].

Space truss A, the top members have suffered from large lateral deformation especially at the corners while the bottom chords suffer from twisting and this can be noticed from **Fig.(10)**.

For space truss B, the top chord members especially those close to the corner diagonals suffer from lateral deformations and this can be noticed from **Fig.(11)**.

6. CONCLUSIONS.

The three-dimensional finite element (ANSYS 11.0) models were used to represent the space truss, top chord members rectangular hollow sections (RHS), bottom chord members, and diagonal members circular hollow sections (CHS), and found to be efficient to simulate these space trusses. In general it can be said that there was good agreement between the analytical and the experimental load-deflection curves at the center of the space truss. Also the finite element models of the space truss simulate the lateral deflection of the top chord members especially at the corners of the truss A and B, and the twisting of the bottom chords. This behavior was reflected in overall ductile performances of the space truss. In other words, the trusses could withstand gradual losses of stiffness induced by several cases of bottom member yielding and top member lateral deformation.

And from the analysis, it can be concluded that the corner parts in the space truss are the most critical parts and their twist are due to lateral twisting of the top chords and this leads to failure of the whole structure. So from **fig.(A-1)** which shows the supports type which are two hinges at two ends and two rollers at the other opposite ends, and from **table(A-1)** in addition to **figs.(A-2 and A-3)** as shown in the **Appendix (A)** below, it can be notice that the maximum deflections in the three axis (x, y and z) occurred at the hinge supports for space truss A, while they occurred in space truss B at the roller support, and this can be justified to the difference in the design between the two trusses A (with overdesigned top chord members (by 30%)) and truss B (with underdesigned top chord members (by 40%)).

7. REFERENCES.

[1] Ahmed El-Sheikh, "Development of a New Space Truss System", Journal Construction of Steel Research, Elsevier, Vol. 37, No. 3 pp. 205-227, 1996.

[2] Henning Agerksov, "Optimum Geometry Design of Double-Layer Space Trusses", Journal of Structural Engineering, Vol. 112, No. 6, June, 1986.

[3] Erling Murtha-Smith, "Alternate Path Analysis of Space Trusses for Progressive Collapse", Journal of Structural Engineering, Vol. 114, No. 9, September, 1988.

[4] Christopher D. Hill, Goerge E. Blandford, and Shein T. Wang, "Post-Buckling Analysis of Steel Space Trusses", Journal of Structural Engineering, Vol. 115, No. 4, April, 1989.

[5] A. I. El-Sheikh and H. El-Bakry, "Experimental Study of Behavior of New Space Truss System", Journal of Structural Engineering, Vol. 122, No. 8, August, 1996.

[6] Omer Kelesoglu and Mehmet ülker, "Fuzzy Optimization Geometrical Space Truss Design", Turkish Journal Engineering Environmental Science, 29, pp. 321-329, 2005.

[7] Jin-Woo Kim, Jeung-Hwan Doh, and Yong-Hee Lee, "Behavior Characteristics of Hyper Shaped Space Truss with Some Removed Members", ASCE, 978-0-7844-1016-5, pg. NA, 2008.

[8] ANSYS, Inc., "ANSYS Help", Release 11.0, Documentation, Copyright 2007.

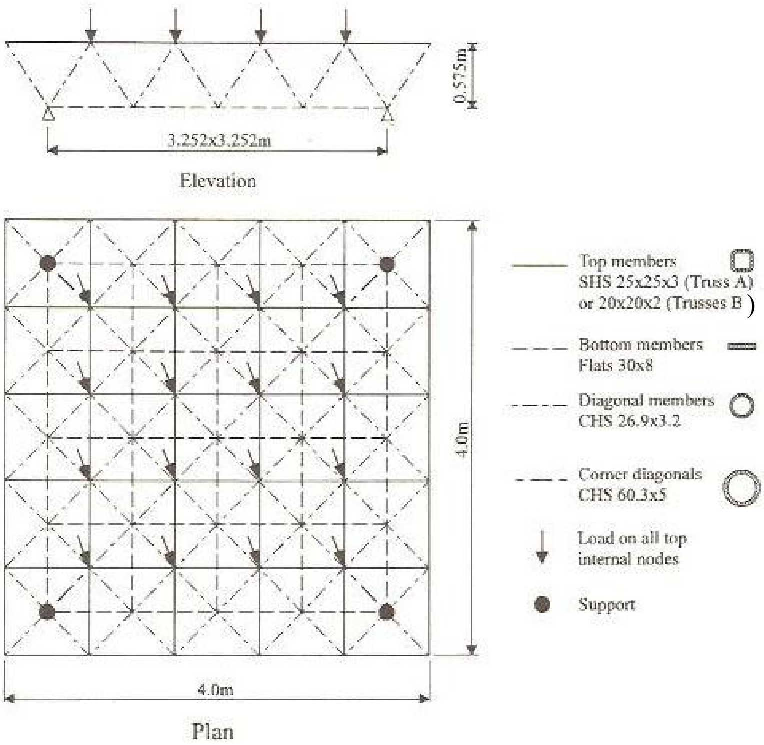
[9] Beer F., Jr E. Russell, and DeWolf J., "Mechanics of Materials", McGraw-Hill, Singapore, 2006.

[10] Roylance D., "Stress-Strain Curves", http://ocw.mit.edu, pp.1-14, 2001.

[11] Plastic Design in Steel A guide and Commentary, ASCE-Manuals and Reports on Engineering Practice, By a Joint Committee of the Welding Research Council and the American Society of Civil Engineers, 1971.

Table(1):Comparison of ultimate loads predicted by ANSYS 11.0 with experimental values.

Specimens	Ultimate Load P_u (kN)		$\frac{(Pu)ANSYS}{(Pu)EXPT}$	Central Deflection Δ_c (mm)		$\frac{(\Delta u)ANSYS}{(\Delta u)EXPT}$
	$(P_u)_{ANSYS}$	$(P_u)_{EXPT}$		$(\Delta_c)_{ANSYS}$	$(\Delta_c)_{EXPT}$	
Truss A	390	390	1	47.45	47	1.01
Truss B	218	218	1	32.65	32	1.02



Figure(1): Layout of test trusses [1].

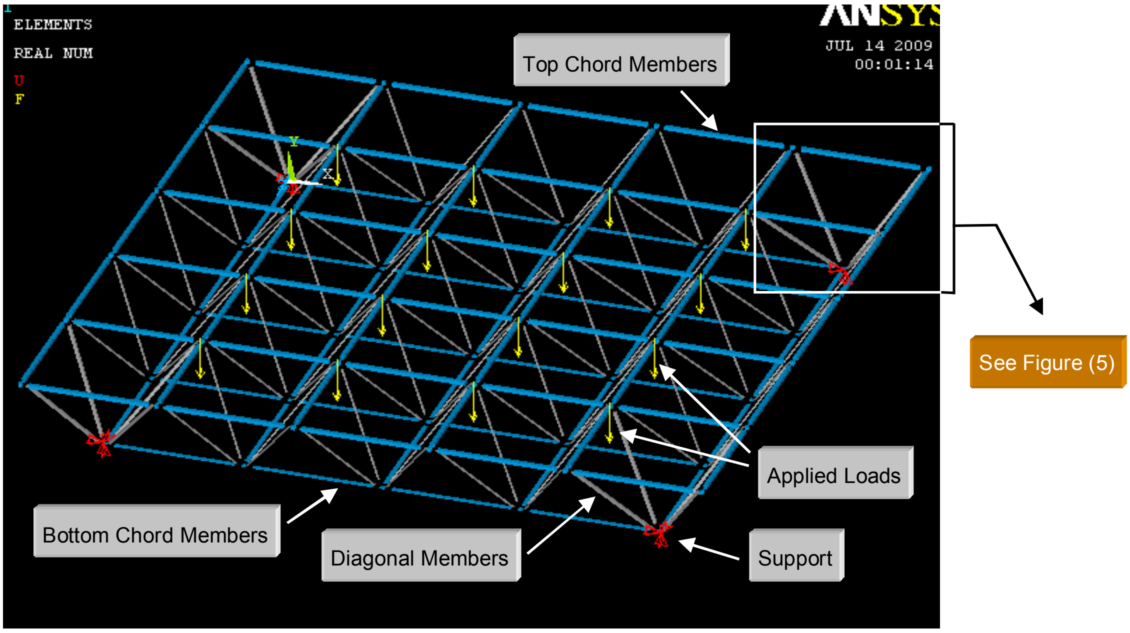
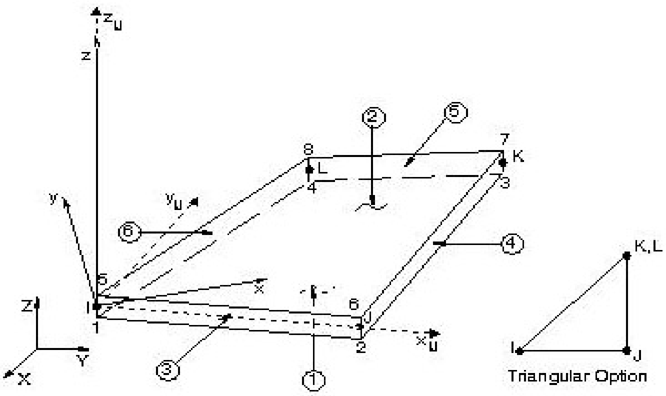


Figure (2): ANSYS mesh of space truss A.



x_{IJ} = Element x-axis if ESYS is not supplied.

x = Element x-axis if ESYS is supplied.

Figure (3): Shell (143) geometry [8].

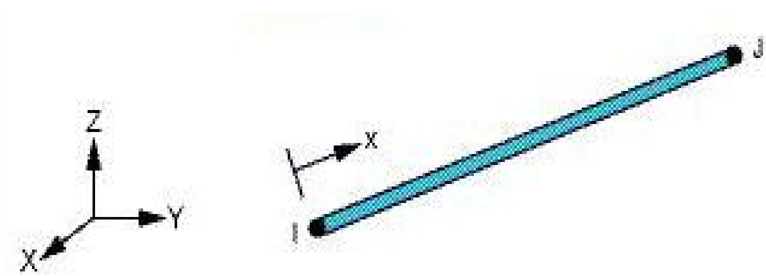


Figure (4): Link-180 geometry [8].

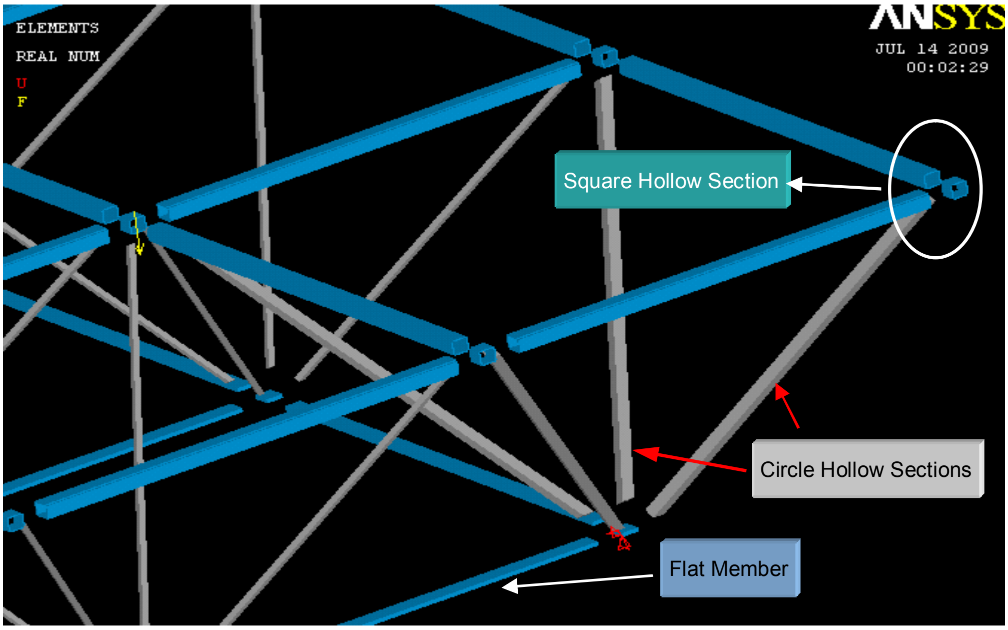
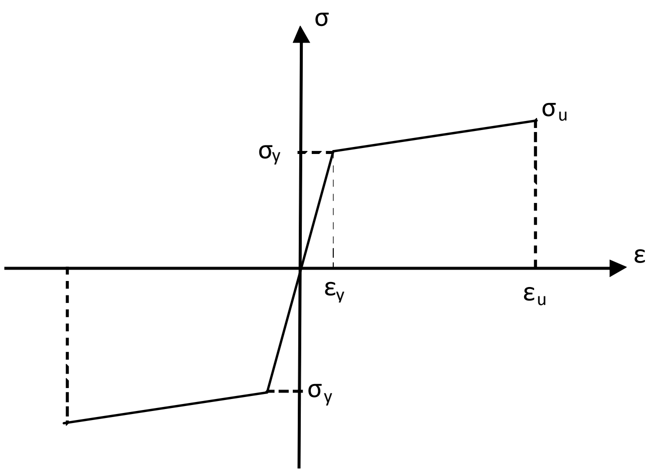
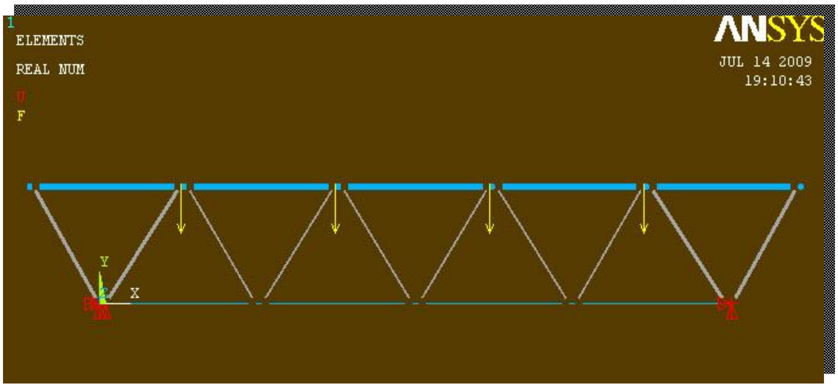


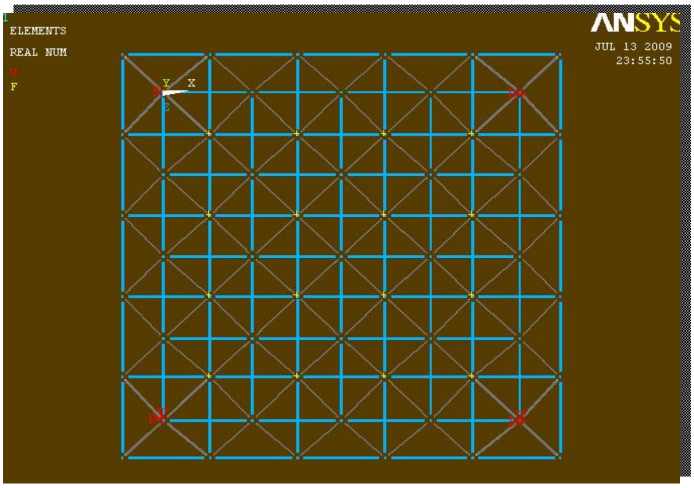
Figure (5): Section of the space truss (ANSYS program 11.0).



Figure(6): Idelized uniaxial stress-strain relationships for steel [8].



(a) Side view of the space truss (ANSYS program 11.0)



(b) Top view of the space truss (ANSYS program 11.0)

Figure (7): Space truss mesh by ANSYS program release 11.0.

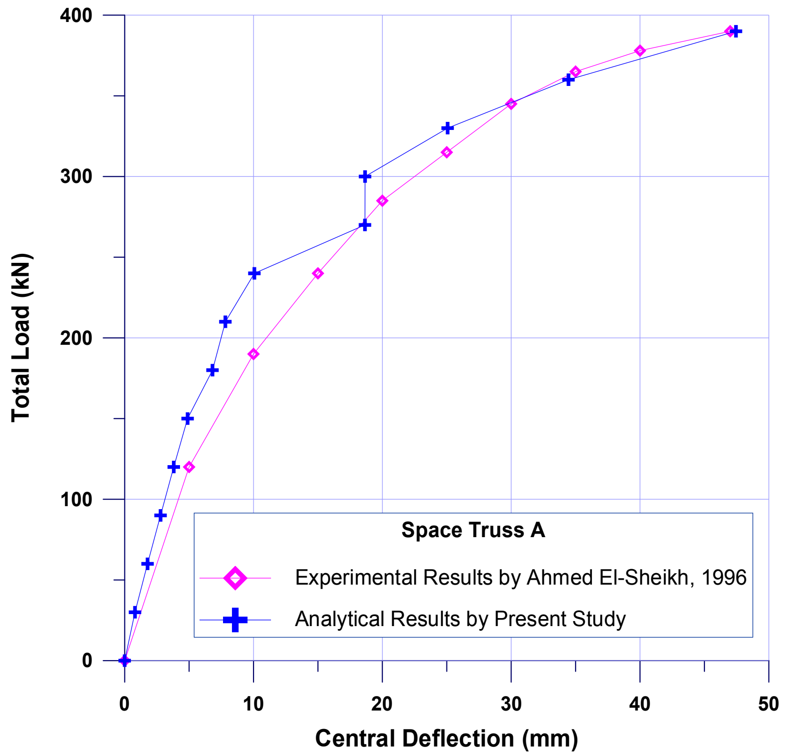


Figure (8): Load-Deflection relationship: analytical-experimental comparison of space truss A.

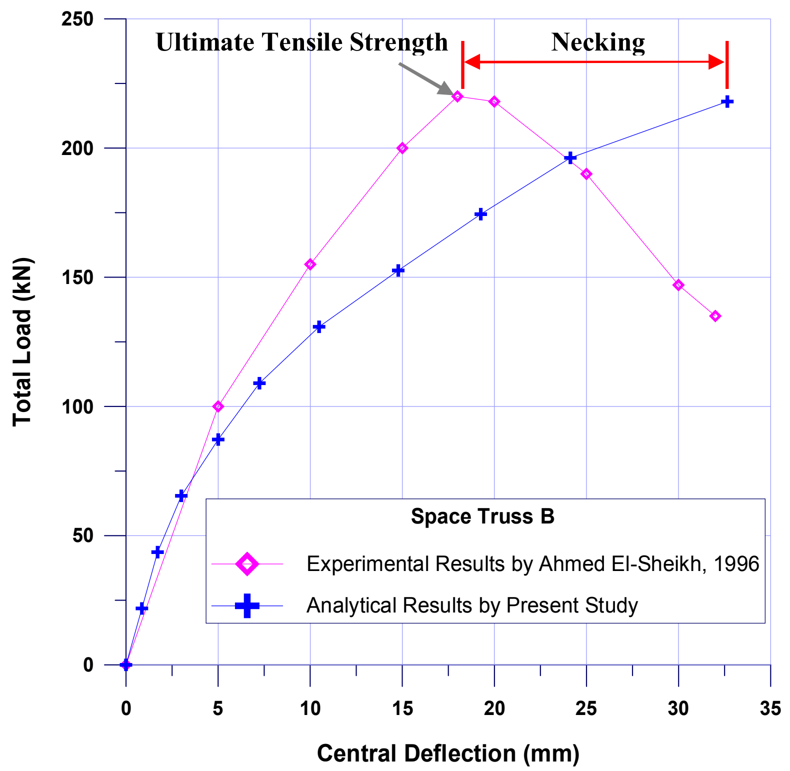


Figure (9): Load-Deflection relationship: analytical-experimental comparison of space truss B.

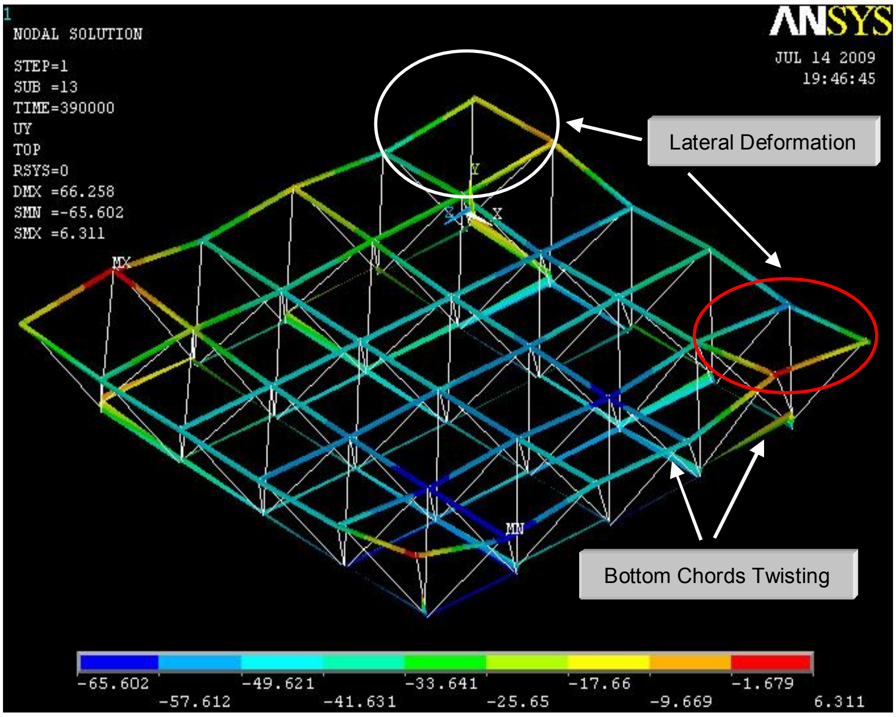


Figure (10): Space truss A, Contour (Y) deformed shape.

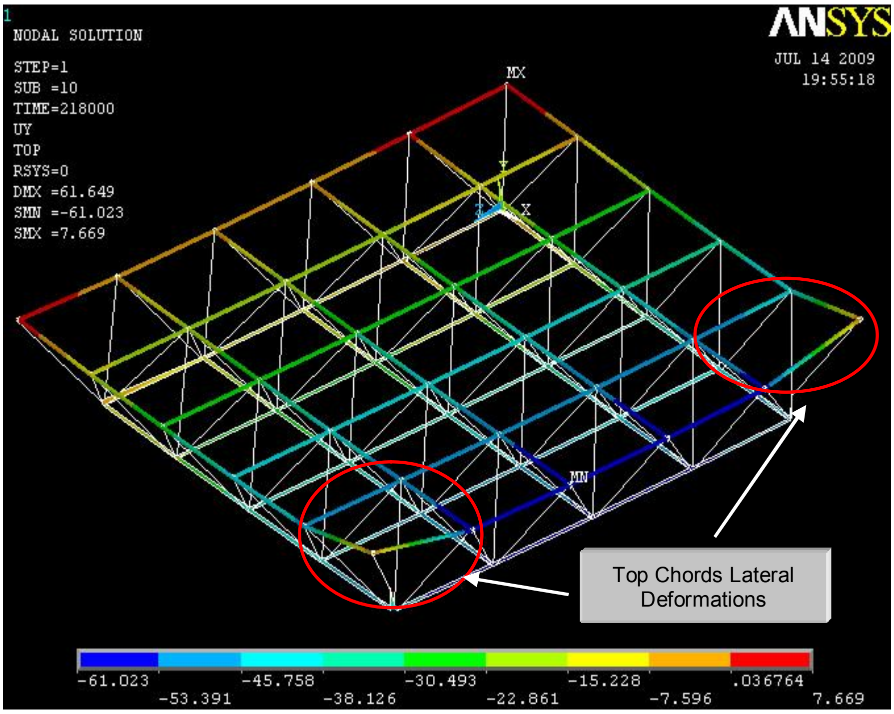


Figure (11): Space truss B, Contour (Y) deformed shape.

Appendix A:

Table(A-1): Maximum nodes deflection at Truss A and B.

Truss A			Truss B		
Ux (mm)	**Uy** (mm)	**Uz** (mm)	**Ux** (mm)	**Uy** (mm)	**Uz** (mm)
13.089 (Top Chord) -3.606 (Top Chord)	6.311 (Top Chord) -65.602 (Top Chord)	7.167 (Top Chord) -1.708 (Bottom Chord)	11.117 (Top Chord) -0.0274 (Bottom Chord)	7.669 (Top Chord) -61.023 (Top Chord)	1.58 (Top Chord) -1.958 (Bottom Chord)

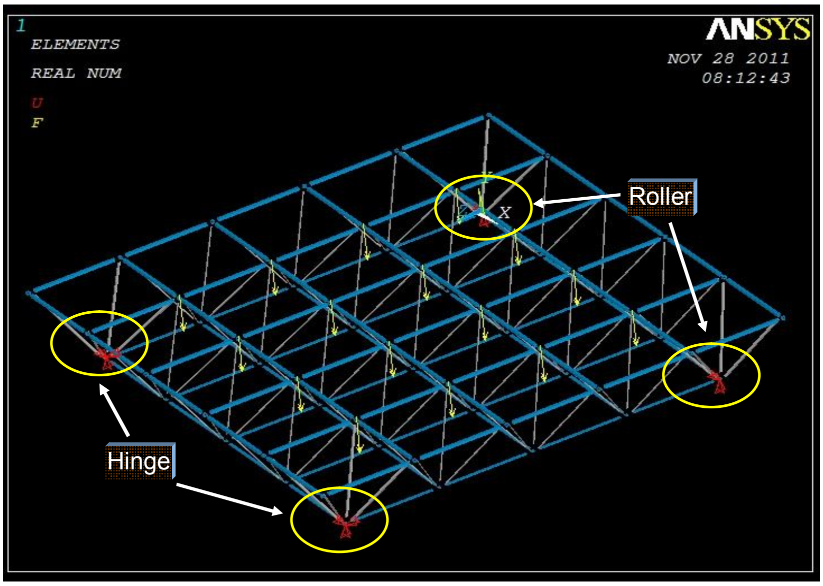
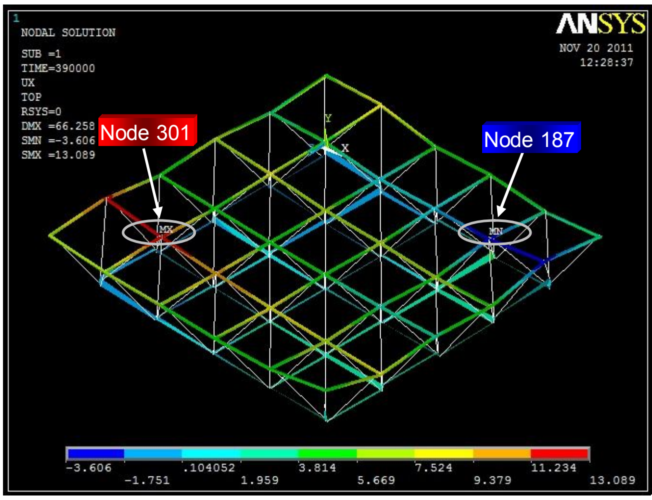
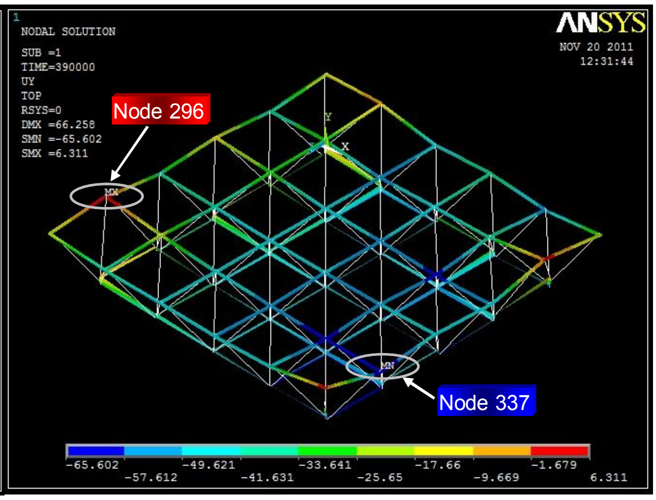


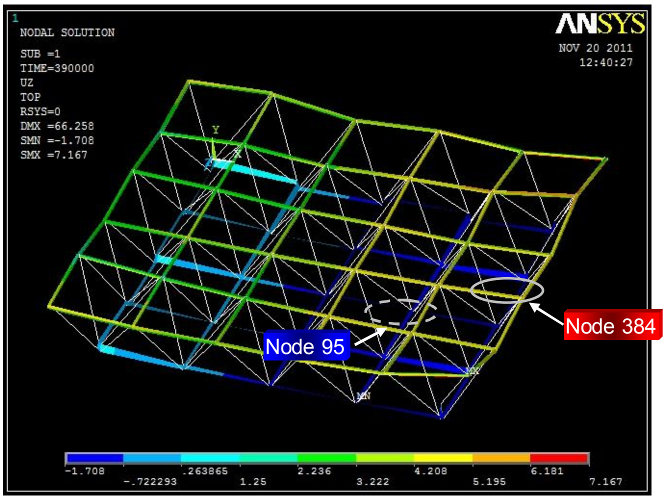
Figure (A-1): Space truss (A) supports by (ANSYS program11.0).



(a) Solution contours results in X-component of displacement for Space Truss A.

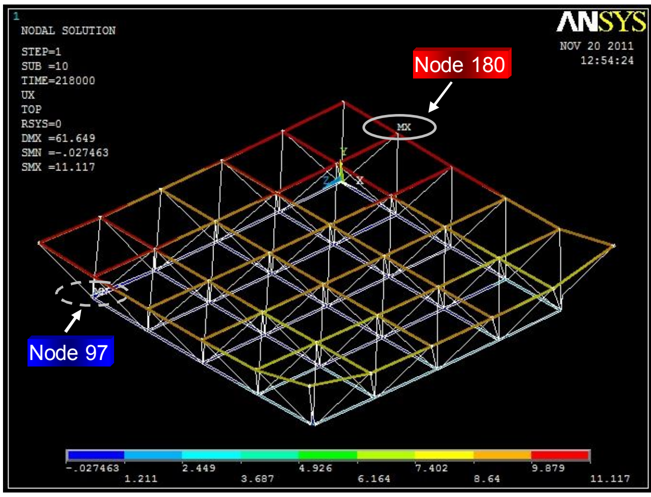


(b) Solution contours results in Y-component of displacement for Space Truss A.

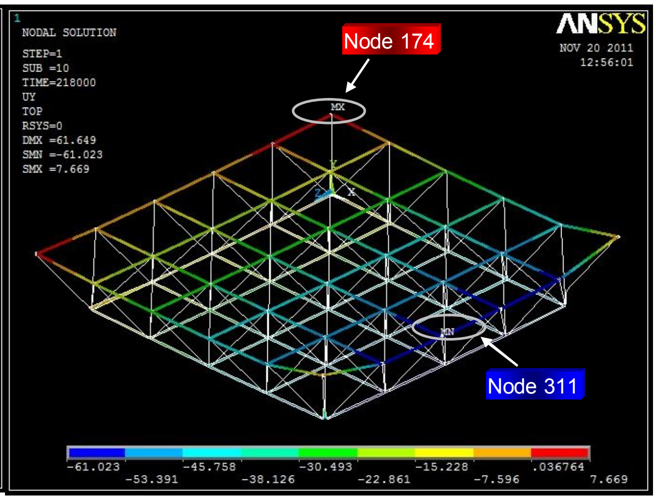


(c) Solution contours results in Z-component of displacement Space Truss A.

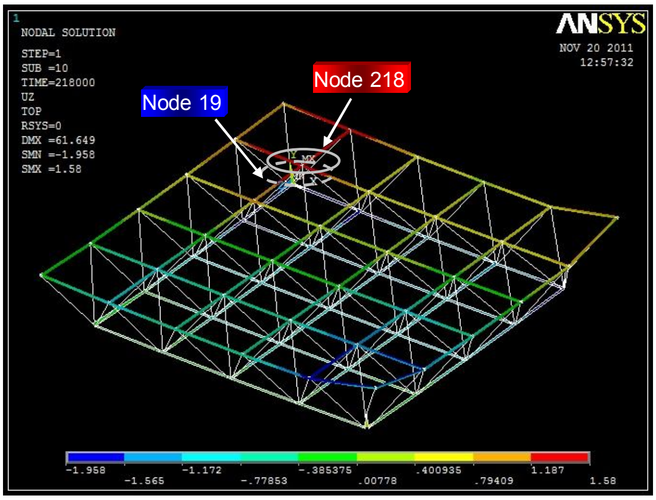
Figure (A-2): Space truss (A) contours deformed shape nodal solution.



(a) Solution contours results in X-component of displacement for Space Truss B.



(b) Solution contours results in Y-component of displacement for Space Truss B.



(c) Solution contours results in Z-component of displacement Space Truss B.

Figure (A-3): Space truss (B) contours deformed shape nodal solution.

التحليل اللاخطي باستخدام العناصر المحددة للمسنمات الفضائية

م.د.أحمد فرحان كاظم
قسم هندسة البناء و الأنشاءات
الجامعة التكنولوجية

الخلاصــة.

يقدم هذا البحث دراسة تحليلية تتضمن استخدام العناصر المحددة ثلاثية الأبعاد لتمثيل مسنم ثلاثي الأبعاد (في الفضاء) باستخدام برنامج (ANSYS 11.0) على الحاسبة ، حيث ان النتائج التحليلية اظهرت تطابق جيد جدا و بنسبة (100%) مع النتائج العملية، في حين ان خيار الرسم في هذا البرنامج عكس تصرف المنشأ تحت تأثير الأحمال المسلطة بسبب قابلية هذا الخيار لتمثيل التصرف الحقيقي للمنشأ تحت هذه الاحمال. كذلك فان نموذج العناصر المحددة للمسنمات الثلاثية الأبعاد مثلت التشوه الجانبي للاعضاء الوترية العليا خاصة عند الزوايا و الألتواء للاعضاء الوترية السفلى، كما كانت أكبر تشوهات في كل من المحاور (x, y , y) تقع عند المساند من نوع (مفصل، hinge) بالنسبة للمسنم (A) أما بالنسبة للمسنم (B) فكانت أكبر تشوهات تقع عند المساند من نوع (بكرة، roller).

الكلمات الرئيسية: الجملون الفضائي،الوتد العلوي،الوتد السفلي،العناصر المحدودة،برنامج ANSYS،عنصر قشري.