Republic of Iraq

Ministry of Higher Education

and Scientific Research

Babylon University

OPTIMUM DESIGN OF

STIFFENED PLATE-STRUCTURE SUBJECTED

TO STATIC LOADING

A Thesis

Submitted to the College of Engineering of the University

of Babylon in Partial Fulfillment of Requirements

for the Degree of Master of Science in

Mechanical Engineering

(Applied Mechanics)

y

MOHAMMED ALI S. ALSHJARY

(B.Sc.)

٠٠٦

جمهورية العراق

وزارة التعليم العالي

والبحث العلمي

جامعة بابل

التصميم المثالي لتراكيب الصفائح المقواه المعرضه إلى حمل ساكن

رسالة

مقدمة إلى كلية الهندسة في جامعة بابل

كجزء من متطلبات نيل درجة ماجستير علوم

في الهندسة الميكانيكية

(ميكانيك تطبيقي)

أعدت من قبل

محمد علي صيهود

بكالوريوس هندسة مكانيك

٢٠٠٦

بِسۡمِ ٱللَّهِ ٱلرَّحۡمَٰنِ ٱلرَّحِيمِ

اِقْرَأ باسْمِ رَبِّكَ الَّذِي خَلَقَ ﴿١﴾ خَلَقَ الإنْسَانَ مِنْ عَلَقٍ
﴿٢﴾ اِقْرأ وَرَبَّك الأكْرَمُ ﴿٣﴾ الَّذِي عَلَّمَ بِالْقَلَمِ ﴿٤﴾ عَلَّمَ
الإنْسَانَ مَالَمْ يَعْلَمْ ﴿٥﴾

صدق الله العلي العظيم

سورة العلق

الآيات (١-٥)

Abstract

The field of structural optimization (optimal design) has grown rapidly over the past decades with many different optimization methods that could be used to produce a structure of minimum weight. This research deals with two aspects, in the first, a general numerical technique based on the finite element analysis and it suggests to investigate the preliminary behavior of metal stiffened plate under action of static load environment. The technique was included a finite element modeling of the structures using high- order isoparimetric plate elements and used to create a certain models to obtain their optimum design. The models are characterized such that, each model is builded using different types of stiffener configuration.

The second aspect was concerned with the investigation of the optimum design configuration of the structures. The optimization techniques that used is called Morphing Evolutionary Structural Optimization (MESO). Which is developed by Xie and Steven in ١٩٩٢.

The Morphing ESO was examined in this research to be applied on stiffened plate structures. The Morphing ESO is based on the simple concept that by slowly removing efficient material from a structure, the residual shape evolves in the direction of making the structure better. The mathematical representation of this method is accomplished in this thesis with full programming and modification required being applicable to a new structure with a new condition. Where the thickness of the plate and stiffeners, and the stiffener height are the design variable. While the objective of the optimization is the structure weight and inequality constraints are the maximum Von Misses stress required for each structure.

From the result obtained , it is found that the maximum Von Misses stress occurs at a distance from ٠-١٥٪ from the plate root and decreases in the direction of the free end, and it is found that the design optimization for rectangular- honeycomb plate structure gives the best result than the other models. All optimization programs have been written using ANSYS Parametric Design Language in order to be used the optimization program together with the ANSYS package capability. Where the APDL was used as a sub- program to analyze the structure and to easily transfer the information between the optimization and the analysis programs.

List of Contents

الخلاصـــــة

أن مجال الامثلية الهيكلية (التصميم المثالي) أخذ يتطور بسرعة خلال العقود الماضية مع العديد من طرق الامثلية المختلفة التي يمكن استعمالها للحصول على تركيب يتميز بوزن أقل.

يتضمن البحث جزئين , الجزء الأول يتضمن اقتراح تقنيه عدديه اعتمدت طريقه العناصر المحددة كأساس لها لتوضيح سلوك الصفائح المقوات والتراكيب الاسطوانية تحت تأثير ظروف الأحمال الساكنة. التقنية تضمنت نمذجة (صياغة) طريقه العناصر المحددة باستخدام عنصر مكافى الخواص (متشابهه الخواص) ذو الأربع عقد ثم ربطت هذه العناصر لخلق نماذج معينة بغيه أيجاد التصميم الأمثل لها. أن هذه النماذج سوف تتميز بذلك وان كل نموذج يستعمل أنواع مختلفة من المقويات.

الجزء الثاني يتضمن توضيح التصميم الامثل للموديل المقترح, حيث أن تقنيه الامثلية المستخدمة تدعى (MESO) التي طورت عن طريق العالمان (Xie) و(Steven) عام ١٩٩٢ م.

هذة الطريقة فحصت في هذا البحث لتطبيقها على الصفائح المقوات وتسند هذه الطريقة على فكره بسيطة وذلك بازاله البطيئة للمادة الغير المؤثرة من التركيب, بحيث ان الشكل المتبقي يتجه نحو تركيب أفضل. ان التمثيل الرياضي لهذه الطريقه تامة في هذا البحث بالبرمجة الكاملة والتعديل المطلوب بحيث تكون قابله للتطبيق على تراكيب أخرى وبشروط جديدة. يمثل سمك الصفيحة وسمك المقويات (honeycomb stiffeners) وارتفاعها هي متغيرات التصميم , إما هدف تحقيق الامثلية فيمثل بوزن التركيب بينما تمثل القيود بإجهاد الفون مايسز (Von Misses).

من النتائج التي تم الحصول عليها , هو ان أعلى إجهاد يوجد عند مسافة (١٥%-٢٠%) من جذر الصفيحة المقوات ويقل ذالك الإجهاد باتجاه النهاية الحرة للصفيحة. كما وجد ان تركيب الصفائح المقوات ذات خلايا النحل المر بعه (Rectangular-honeycomb cells) تعطي نتائج أفضل من التراكيب الأخرى, وان تركيب السندوتش الاسطواني(Sandwich Cylindrical Structural) يعطي نتائج أفضل من التركيب الاسطواني ذات خلايا النحل المر بعه.

جميع برامج الامثلية كتبت باستخدام (ANSYS Parametric Design Language) (APDL) حيث يستخدم هذا البرنامج سويا مع(ANSYS) ويمثل كونه برنامج فرعي يمكن بواسطة تحليل التركيب ونقل المعلومات بسهوله بين تحقيق الامثلية وبرامج التحليل.

ACKNOWLEDGMENTS

"Praise be to ALLAH, Lord of the whole creations"

I would like to sincerely and wholeheartedly thank Dr. Hatem H. Al-Taee and Dr. Adil A. AL-Moosawy for their guidance, kindness, and their patience as advisors, boundless energy while teaching, promptness while reviewing all my writing and passion for research in this work. I am indebted to them for helping me to achieve the research and thus opening a new exciting world for me.

Most importantly, I would like to thank my parents, for their unconditional support, love, and affection. Their encouragement and never ending kindness made everything easier to achieve.

Thanks to the mechanical department and college of engineering of Babylon University.

Finally, I apologize for all those whom I forget to mention their names.

MOHAMMED ALI SAIHOOD

٢٠٠٦

CERTIFICATION

*We certify that this thesis entitled **"OPTIMUM DESIGN OF STIFFENED PLATE- STRUCTURE SUBJECTED TO STATIC LOADING "** was prepared by **MOHAMMED ALI SAIHOOD** under our supervision at the University of Babylon in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering (Applied Mechanics).*

Signature:	Signature:
Name: **Dr. Hatem H. Al-Taee**	Name: **Dr. Adil A. AL-Moosawy**
Lecture	Assist. Professor

EXAMINING COMMITTEE CERTIFICATE

We certify that we have read this thesis entitled **" Optimum Design of Stiffened Plate- Structure Subjected to Static Loading"** and as an examining committee, examined the student , **" Mohammed Ali Saihood"** , in its contents and that in our opinion it meets standard of a thesis for the degree of Master of Science in Mechanical Engineering.

Signature:
Name: Lecture
Dr. Hatem H. AL- Taee
(Supervisor)
Data: / /٢٠٠٦

Signature:
Name: Asst. Prof.
Dr. Adil A. AL- Moosawy
(Supervisor)
Data: / /٢٠٠٦

Signature:
Name: Lecture
Dr. Basim A. AL-Maksosi
(Member)
Data: / /٢٠٠٦

Signature:
Name: Asst. Prof.
Dr. Alaa'.M.AL-Jesany
(Member)
Data: / /٢٠٠٦

Signature:
Name: Asst. Prof.
Dr. Ahamed A. AL-Rajihy
(Chairman)
Data: / /٢٠٠٦

Approval of the Mechanical Engineering Department.
Head of the Mechanical Engineering Department.

Signature:
Name: Asst. Prof.
Dr. Adil A. AL- Moosawy

Data: / /٢٠٠٦

Approval of the College of Engineering
Dean of the College of Engineering.

Signature:
Name: Prof.
Dr. Abdul AL-Wahid K.Rajih
Data: / /٢٠٠٦

Dedication

To...

My Family

and My Friends

MOHAMMED ALI SAIHOOD
2006

الإهداء

لو أنكرت الثمرةُ الورقة ......

فلقد أنكرت أصلها .... وظلمتْ بختها

فعرفاناً بالجميل:

للورقةِ والغصنِ .... أمي وأبي

للجذعِ والجذرِ .... الإسلام المحمدي

للماءِ والهواءِ .... أساتذتي

منكم تعلمتُ .... أنْ أُطأطأُ رأسي تواضعاً,

وأنْ أرفعهُ باحثاً متسائلاً !!

جُلّ أحترامي ... وتقديري ...... والسلام .

محمد علي صيهود

NOTATIONS

Symbol	Meaning	Unit
A	Cross section area	m^2
E	Elasticity modulus of isotropic material	N/m^2
[E]	Elasticity matrix	N/m^2
f	Shear correction factor	---
{F}	Overall load vector	N
G	Shear (rigidity) modulus of isotropic material	N/m^2
$\mathbf{h_s}$	Height of stiffeners	mm

[K]	Overall stiffness matrix	N/m
$[k]^e$	Element stiffness matrix	N/m
L_1	Length of finite element model along global X-axis	mm
L_2	Length of finite element model along global Y-axis	mm
l_s	Side length of rectangular or diagonal cell	mm
Ni(ξ,η)	Shape function at node i	----
PI	Performance index	----
RR	Rejection ratio	----
S.F	Safety factor	----
SS	Steady state number	----
t	Thickness	mm
t_p	Thickness of plate	mm
t_s	Thickness of stiffeners	mm
X,Y,Z	Global coordinate system axis	----
x,y,z	Nodal coordinate system axis	----
$\{u\}^e$	Element displacement vector	mm
u_i	Linear displacement along element x-axis	mm
v_i	Linear displacement along element Y-axis	mm
w_s	Width of quadrilateral element	mm
w_i	Linear displacement about element Z-axis	mm
θ_{xi}	Angular displacement about element X-axis	rad
θ_{yi}	Angular displacement about element Y-axis	rad
σ_i	Normal stress in i-direction	N/m^2
ε_i	Normal strain in i-direction	----

$\{\varepsilon\}$	Strain vector	----
$\{\sigma\}$	Stress vector	----
τ_{ij}	Shear stress component through ij-plane	N/m٢
v	Poisson’s ratio of isotropic material	----
ξ,η,ς	Intrinsic coordinate system	----
λ_i	Lagrange multipliers	----
[ ]-١	Inverse matrix	----

Note :-

Other symbols are given in their corresponding chapters.

"Appendix-B"

Elastic Failure Criterion Theories

B.١ Maximum-shear-stress theory:

This theory assumes that failure(yielding or fracture) occurs for a combined stress condition when the maximum shear stress equals the value of a critical shear stress(yield shear stress or ultimate shear stress) produced in an element subjected to simple tension. For a three- dimensional stress state, the maximum shear stress are given by one of the following expressions:

$$\frac{\sigma_2-\sigma_3}{2}, \quad \frac{\sigma_3-\sigma_1}{2}, \quad \frac{\sigma_1-\sigma_2}{2}, \qquad ...(B\text{-}١)$$

Where $\sigma_1, \sigma_2, \sigma_3$ are the principal stresses. In the case of an element subjected to simple tension at the yield condition, it can be shown that:

$$\tau_y = \frac{\sigma_y}{2} \quad \text{...(B-٢)}$$

Where σ_y, τ_y are the tensile and shear yield stresses. In accordance with the maximum- shear- stress theory, yielding will occur where the largest shear stress in the element (given by one of the terms in Eq.(B-١))reaches a vale equal to the shear yield stress τ_y in simple tension. If it is assumed that the shear stress at yield for simple tension is equal to that for simple compression, then failure occurred at one of the following:

$$\left.\begin{aligned} \sigma_1 - \sigma_2 &= \pm\sigma_y \\ \sigma_2 - \sigma_3 &= \pm\sigma_y \\ \sigma_3 - \sigma_1 &= \pm\sigma_y \end{aligned}\right\} \quad \text{...(B-٣)}$$

Where the $\pm$ sign means that the maximum shear stress may be positive or negative.

The maximum-shear- stress theory is widely used by designers for predicting failure of ductile materials by yielding.

B.٢ Von-Misses (Distortion-energy theory):

This theory is concerned mainly with predicting the beginning of yielding. However, in accordance with this criterion, yielding will occur when the strain energy of distortion per unit volume equals the strain energy of distortion per unit volume for specimen in uni-axial tension or compression (strained to the yield stress).

The strain energy of distortion per unit volume U_s for a body subjected to a tri-axial state of stress is given by:

$$U_s=\frac{1+\upsilon}{6E}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right] \quad \text{...(B-٤)}$$

Yielding would occur when:

$$\frac{1+\upsilon}{6E}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]=\frac{1+\upsilon}{6E}*2\sigma_y^2$$

or

$$(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2=\text{٢}\,\sigma_y^2 \quad \text{...(B-٥)}$$

For a bi-axial-stress state (σ_3=٠), the distortion-energy criterion for yieling can be expressed by:

$$\sigma_1^2-\sigma_1\sigma_2-\sigma_2^2=\sigma_y^2 \quad \text{...(B-٦)}$$

This equation is that of an ellipse whose major semi-axis is $\sqrt{2}\,\sigma_y$ and minor semi- axis is $\sqrt{2/3}\,\sigma_y$.

Experiments have shown that the distortion- energy theory is in better agreement with data from the yielding of bodies under combined stress. In general, the distortion- energy theory is recommended for defining yielding of ductile materials; however, the maximum- shear theory is a good approximation. In applying the distortion- energy theory for design purpose, a criteria value extracted from Eq.(B-٥) to compared with tensile yield stress of the material, this criteria value called Von-Misses stress σ_{von}:

$$...(B\text{-}٦)\ \sigma_{von} = \frac{1}{2}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$$

Appendix "A"

A.١ Shape Function :

A.١.١: First order (Linear) plate elements(*KOKO,T.S.,*)[٤]

-For quadrilateral elements:

$$N_{١}(\xi,\eta)=\frac{1}{4}(١-\xi)(١-\eta)$$

$$N_{٢}(\xi,\eta)=\frac{1}{4}(١+\xi)(١-\eta)$$

$$N_{٣}(\xi,\eta)=\frac{1}{4}(١+\xi)(١+\eta)$$

$$N_{٤}(\xi,\eta)=\frac{1}{4}(١-\xi)(١+\eta)$$

-For Triangular elements:

$$N_{١}(\xi,\eta)=\xi(٢\xi-١)$$

$$N_{٢}(\xi,\eta)=\eta(٢\eta-١)$$

$$N_{٣}(\xi,\eta)=(١-\xi-\eta)(١-٢\xi-٢\eta)$$

A.١.٢ Second order (Quadratic) plate elements (*KOKO,T.S.,*)[٤]

-For quadrilateral elements:

$$N_{١}(\xi,\eta)=-\frac{1}{4}(١-\xi)(١-\eta)(١+\xi+\eta)$$

$$N_{٢}(\xi,\eta)=-\frac{1}{4}(١+\xi)(١-\eta)(١-\xi+\eta)$$

$$N_{٣}(\xi,\eta)=-\frac{1}{4}(١+\xi)(١+\eta)(١-\xi-\eta)$$

$$N_{٤}(\xi,\eta)=-\frac{1}{4}(١-\xi)(١+\eta)(١+\xi-\eta)$$

$$N_{٥}(\xi,\eta)=\frac{1}{2}(١-\xi^{٢})(١-\eta)$$

$$N_{٦}(\xi,\eta)=\frac{1}{2}(١+\xi)(١-\eta^{٢})$$

$$N_{٧}(\xi,\eta)=\frac{1}{2}(١-\xi^{٢})(١+\eta)$$

$$N_{٨}(\xi,\eta)=\frac{1}{2}(١-\xi)(١-\eta^{٢})$$

-For Triangular elements:

$$N_{١}(\xi,\eta)=\xi(٢\xi-١)$$

$$N_{٢}(\xi,\eta)=\eta(٢\eta-١)$$

$$N_{٣}(\xi,\eta)=(١-\xi-\eta)(١+٢\xi-٢\eta)$$

$$N_{٤}(\xi,\eta)=٤\xi\eta$$

$N_{٥}(\xi,\eta)=٤\eta\,(١-\xi-\eta)$

$N_{٦}(\xi,\eta)=٤\xi\,(١-\xi-\eta)$

A.٢ Shear Correction Factor:

In ***Timshenko's*** deep beam theory, the transverse shearing stress is assumed constant. Actually, the transverse shear stress is zero at the top and bottom sides of the section and maximum at the neutral axis (usually parabola for rectangular sections).

To attain the parabolic shear stress distribution, the equality concept of the strain energy in both constant and parabolic distributions is used. The shearing strain energy for an element (dx dy dz), shown in Figure (A.١), under shearing stress is:

$$\Omega = \int_{Vol} \frac{1}{2}\,\gamma\,\tau\, d(Vol) = \int_{Vol} \frac{\tau^2}{2G}\, d(Vol) \qquad ...(A.١)$$

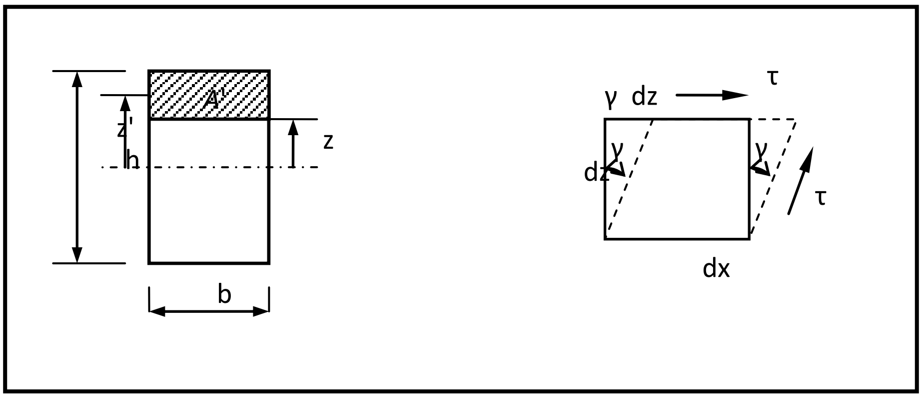


Figure (A.١): Beam element undergoing shear deformation.

For a constant shearing stress distribution, the strain energy is equal to:

$$\Omega_1 = \int_{Vol} \frac{\tau^2}{2KG}\, d(Vol) \qquad \text{...(A.٢)}$$

where K is the shear correction factor.

Now by assuming a piece of beam of unit length with rectangular cross section, equation (A.٢) will reduce to:

$$\text{(A.١)} = \frac{Q^2}{2KGbh} \qquad \text{... (A.٣)}\ \Omega_1 = \frac{Q^2}{2KGA^2}$$

For the parabolic shearing stress, the following relations are used:

$$\text{... (A.٤)} \qquad \tau = \frac{QA'z'}{Ib}$$

$$A' = b\left(\frac{h}{2} - z\right)$$

$$\text{...(A.٥)} \quad z' = \frac{1}{2}\left(\frac{h}{2} + z\right)$$

... (A.٦)

Substituting Equations (A.٦) and (A.٧) in Equation (A.٥) to find that,

$$\text{... (A.٧)}\ \tau = \frac{6Q(\frac{h^2}{4} - z^2)}{bh^3}$$

then

$$d(Vol) = \frac{36Q^2}{2Gb^2h^2}\left(\frac{h^2}{4} - z^2\right)(1.b.dz) \qquad \Omega_2 = \int_{Vol} \frac{\tau^2}{2G}$$

$$= \frac{18Q^2}{bh^6G}\left\{ ٢ * \int_0^{h/2} \left(\frac{h^2}{4} - z^2\right) dz \right\}$$

$$=\frac{3}{5}(\frac{Q^2}{bhG}) \qquad \text{... (A.٨)}$$

Put $\Omega_2 = \Omega_1$

$$\frac{3}{5}(\frac{Q^2}{bhG}) = \frac{Q^2}{2KGbh} \qquad K=\frac{5}{6}= ١/ f$$ *(Timoshenko, S .,)*[٤١].

A. ٣ Applied Load

A. ٣. ١ Applied Load for plate structure:

Load will be applied on nodes (as nodal force). Nodal force that corresponding to each node is illustrated in Table (A-١).

Node number	Force type	Force (N)	Node number	Force type	Force (N)	Node number	Force type	Force (N)
٤	Fz	-٣١.٧٦٢	٥١	Fz	-٤٧.٦٤٢	٣٢	Fy	-٦٣.٥٢
٦	Fz	-٤٢.٣٥	٦٥	Fz	-٦٣.٥٢٥	٣٢	Fz	-٦٣.٥٢
٨	Fz	-٥٢.٩٣٥	٧٩	Fz	-٧٩.٤٠٦	٣٠	Fy	-٨٤.٧
١٠	Fz	-٦٣.٥٢	٩٣	Fz	-٩٥.٢٨٧	٣٠	Fz	-٨٤.٧
١٢	Fz	-٧٤.١١	١٠٧	Fz	-١١١.١٦	٢٨	Fy	-١٠٥.٨
٢	Fz	-٨٤.٧	١٨	Fz	-١٢٧.٠٥	٢٨	Fz	-١٠٥.٨
٢	Fx	٨٤.٧	١٨	Fx	١٢٧.٠٥	٢٦	Fy	-١٢٧.٠
٤٩	Fz	-٣٩.٧٠٢	٥٣	Fz	-٥٥.٥٨٤	٢٦	Fz	-١٢٧.٠
٦٣	Fz	-٥٢.٩٣٧	٦٧	Fz	-٧٤.١١٢	٢٤	Fy	-١٤٨.٢
٧٧	Fz	-٦٦.١٧١	٨١	Fz	-٩٢.٦٤	٢٤	Fz	-١٤٨.٢
٩١	Fz	-٧٩.٤٠٦	٩٥	Fz	-١١١.١٦	١٤	Fy	-١٦٩.٤

١٠٥	Fz	- ٩٢.٦٤٠	١٠٩	Fz	-١٢٩.٥٨	١٤	Fz	- ١٦٩.٤
١٦	Fz	- ١٠٥.٨٧	٢٠	Fz	-١٤٨	١٤	Fx	- ١٦٩.٤
١٦	Fx	١٠٥.٨٧	٢٠	Fx	١٤٨			

Table (A-١): Nodal forces and corresponding nodes numbers

CHAPTER ONE

Introduction

١.١ Structural Optimization:

Structural optimization has received ever increasing in civil, chemical and especially aeronautical engineering with the advent of high speed computers; the tools of structural optimization are no longer resituated to the classical differential calculus and variation calculus. Indeed, various numerical search techniques have been developed over the past three decades.

The basic idea behind intuitive or indirect design in engineering is the memory of past experiences, subconscious motives, incomplete logical processes, random selections or sometimes mere superstition. This, in general, will not lead to the best design.

The short comings of the indirect design can be overcome by adopting a direct or optimal design procedure. The feature of the optimal design is that it consists of only logical decisions. In making a logical decision, one sets out the constraints and then minimizes or maximizes the objective function (which could be either cost, weight or merit function).

The optimization problem is classified on the basis of nature of equations with respect to design variables. If the objective function and the constraints involving the design variable are linear then the optimization is termed as linear optimization problem. If even one of them is nonlinear it is classified as the non-linear optimization problem. In general the design variables are real but some times they could be integers for example, number of layers, orientation angle, etc. The behavior constraints could be equality constraints or inequality constraints depending on the nature of the problem.

The structural optimization problem can be posed as:

Minimize or Maximize

$F = F(x_1, x_2, x_3, \ldots\ldots\ldots\ldots, x_n)$

Subject to:

$C_1 = C_1\ (x_1, x_2, x_3, \ldots\ldots\ldots\ldots, x_n)$

$$C_{٢}=C_{٢}(x_{١},x_{٢},x_{٣},..........,x_n) \qquad ...(١-١)$$

$$C_n=C_n(x_{١},x_{٢},x_{٣},..........,x_n)$$

And

$$\phi_{١}=\phi_{١}(x_{١}, x_{٢}, x_{٣},..............,x_n) \geq ٠$$

$$\phi_{٢}=\phi_{٢}(x_{١}, x_{٢}, x_{٣},............. ,x_n) \geq ٠ \qquad ... (١-٢)$$

$$\phi_n=\phi_n(x_{١}, x_{٢},x_{٣},...............,x_n) \geq ٠$$

$x_{١},x_{٢},x_{٣},...,x_n$ are the design variables, $C_{١},C_{٢},.......,C_n$ are equality constraints and $\phi_{١}$, $\phi_{٢}$,........, ϕ_n are inequality constraints. The nature of the mathematical programming problem depends on the functional from of F, C, and ϕ, if these are linear function of design variables, and then the mathematical programming problem is treated as linear programming problem. On the other hand if any one of them is a nonlinear function of the design variable, then it is classified as nonlinear programming problem.

There are three main classes of structural optimization problems depending on the type of the design variables used: sizing, shape, and topology. In sizing optimization problems, the aim is usually to minimize the weight of the structure under certain behavioral constraints on stresses and displacements. The design variables are most frequently chosen to be dimensions of the cross-sectional area of the members of the structure. In structural shape optimization problems, the aim is to improve the performance of the structure by modifying its shape. The design variables are either some of coordinates of the key points in the boundary of the structure or some other parameters that influence the shape of the structure. Structural topology optimization assists the designer to define the type of structure that is best suited to satisfy the operating conditions for the problem at hand.

In addition to the three main classes of optimization problem, any combination of them can be implemented. For all three types of design variables have been combined. The aim is to minimize the weight of the structure under certain behavioral constraints. Sizing design variables are related to the definition of the cross section of the stiffeners and the thickness of the plate. The shape design variables control the inclination of the curved surface at the supports, whereas the topology design variables are related to the number and the position of the stiffeners in both the longitudinal and transverse directions. These design variables may be active or nonnative, leading to a combined sizing-shape-topology optimization problem, when all design variables are active or to sizing- shape-topology optimization problem. When only the sizing and shape design variables are active

During the past three decades, many numerical methods have been developed meet the demands of structural design optimization. These methods can be classified in two general categories[١]:

١. Deterministic (Gradient based method).

٢. Probabilistic (Heuristic based method).

Mathematical programming methods, and in particular the gradient based optimizers that have been basically used for solving structural optimization problems in the past, belong to the first category of optimization algorithms. These methods make use of local curvature information, derived from linearization of the original function by using their derivatives with respect to the design variables at points obtained in the process of optimization to construct an approximate model of the initial problem.

In ١٩٩٢, a new method of structural optimization was developed by **(XIE and Steven.,١٩٩٨) [١]** called the Evolutionary Structural Optimization (ESO) method. Evolutionary Structural Optimization (ESO) is a design method based on the simple concept of gradually removing inefficient material from a structure as it is being designed. Through this method, the resulting structure will evolve toward its optimum shape. An engineer must specify the design domain and loads and kinematics constrains. The past research has shown that the ESO method could be successfully applied to all types of elements, i.e. beam, plates and bricks, structural with multiple load cases, to structural dynamic problem and to structure with non-linear properties. However, the ESO method so far, does not allow to incorporate any non- structural constraints to be incorporated during its process **[٢]**.

The initial stages of development of the ESO method were employed in verifying the classical single load problems to demonstrate its applicability. Once the method had been shown to work accurately and efficiently **[١]**, it was then extended to structures with multiply load cases.

١.٢ Analysis of stiffened structures:

The use of stiffened structural elements in most branches of structural engineering began in the nineteenth century with the application of steel flat or curved plates for hulls of ships and subsequently with the development of steel bridges and aircraft structures and other situations where the reduction of self-weight is an important design objective for satisfying the requirements of increased stiffness and reduced weight.

Stiffened plate and cylindrical structures have found widespread in a variety of engineering structures such as steel chimneys, pipes and conduits, missile bodies, side shells of ships, its deck and superstructures, submarines and offshore structures because it can achieve economy in weight with no sacrifice of strength. Stiffened cylindrical structure are very common in engineering practice because they combine high stiffening characteristic with low material volume **[٣]**. Plates stiffened by longitudinal and transverse members are one of the most common structural components. Use of stiffeners makes it possible to resist highly directional loads, and to introduce multiple load paths that many provide protection against damage and crack growth under both compressive and tensile loads. The biggest advantage of the stiffeners though, is the increased bending stiffness of the panel with a minimum of additional material.

Stiffened plates have been considered for used in these weight-sensitive structures, where high strength-to-weight and stiffness-to-weight ratios are required. Besides their high strength and stiffness, stiffened plates are usually thin. Thus, bending is a critical consideration for the optimum design of structures made of such plates. These plates are fabricated as an assembly of individual plates. This allows the designer to select the most effective disposition of material in the cross section to carry the specified loading **[٤]**.

The stiffener is modeled as a separate element and the mathematical formulation is done by considering the stress resultants as the axial force, the bending moment in the plane of its bending and the torsional moment. The stiffener has been modeled employing the same displacement polynomials of the basic shell element which facilitates the formulation of the stiffness matrix of the stiffener in terms of nodal parameters of the shell element.

Fig(١-١) shows types of honeycomb core.

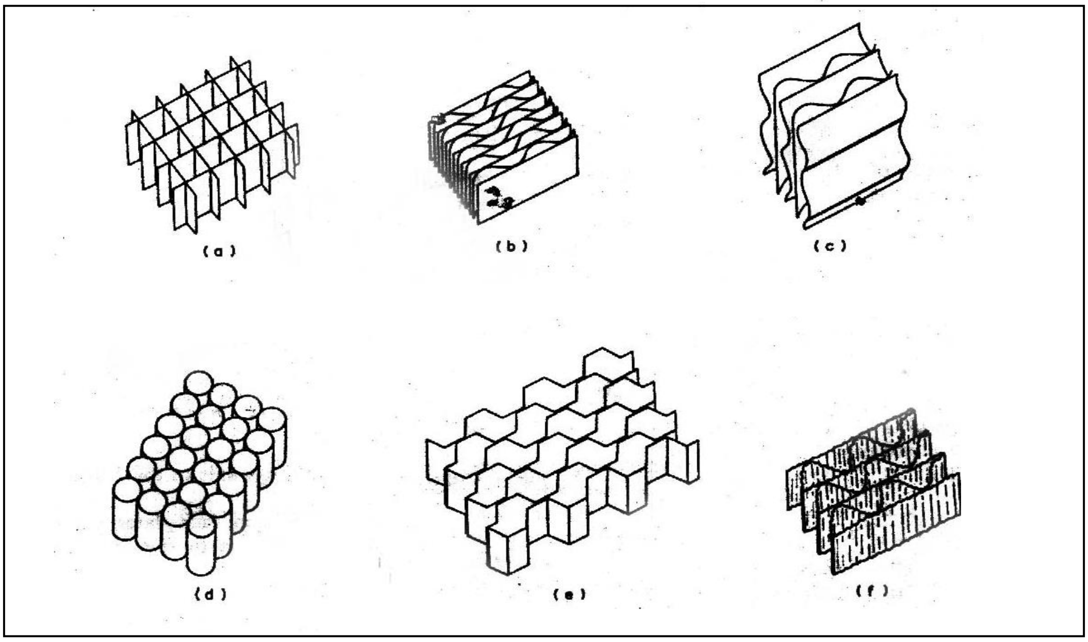


Fig (١-١):Types of honeycomb core

١.٣ Scope of the present work:

The present work has two main aims, namely:

١. To present a better understanding of the behaviour of different types of stiffened plate. A theoretical work using a finite element analysis is carried out to study the behaviour of metal stiffened plates by using a isoparimetric plate element.

٢. To investigate the optimal design of stiffened plates structures. The structural optimization of isotropic stiffened plate structure, the Morphing Evolutionary structural optimization (MESO) method is used while the driving criterion used in this method is the Von Misses stress.

CHAPTER TWO

Literature Review

٢.١ Introduction

This chapter presents a review to some of the literatures that are concerned with the objective of the current work, in addition to the remarks that are extracted from the literature. Only some literatures do not have any new mark, these are explained in the later chapters.

٢.٢ Structure optimization of plates:

***Agarwal* et al [٦]:** determined optimum hat-stiffened compression panel designs by using a structural synthesis technique. Effects of simplifying assumptions made in the bending analysis for the optimization program are investigated using a more accurate analysis, which is a linked plate element program. Optimization results for an aluminum panel are compared with available results. Optimization results for hat-stiffened graphite-epoxy panels show a ٥٠-persent weight savings over optimized aluminum panels. Using the structural synthesis technique, composite panels are shown to posses a variety of proportions at nearly constant weight.

Patnaik. and Sannaran. [٧]: Presented the optimum design of stiffened cylindrical panels weight as the objective function and constraints or the frequencies in the presence of initial stresses by using unconstrained minimization techniques of mathematical programming problem. The interaction between the buckling constraints and the frequency constraints in the presence of initial stresses is inclined in the following. Loss of load carrying capacity due to imperfection and due to suddenly applied load is included in the buckling analysis.

Govil. et al [٨]: Presented the optimum design method with sub structuring to complex wing- type structures. The optimal design problem for idealized wing structures is defined as follows: find the design variable associated with each element of a wing structures. Such that the total weight is minimized, the equations of equilibrium hold and constraints on stress, displacement and member size are satisfied. Design variables for wing structures are taken as cross-sectional areas of truss elements and thickness.

Chang. Lin et al [٩]: presented a general redesign approach that includes the usual stress, displacement, member size and buckling constraints. The layout of the structure is not changed during the design procedure. The optimality criterion derived for all the constraints imposed on the structure is equivalent to the Kuhn-Tucker conditions of nonlinear mathematical programming .However the criterion approach is formulated in a general form with all the constraints being combined together or in a simple form with a single constraint. An optimization computer code ARS٤ (Automatic Resizing System ٤) has been developed and applied to a triple- span wing structure and an aerobatic wing structure.

Rao. and Reddly. [١٠]: considered the design optimization of axially loaded, simply supported stiffened conical shells of minimum weight. The design variables are thickness of shell wall, thickness and depths of rings and stringers, number/spacing of rings and stringers. Overall buckling strength and direct stress constraints are considered in the design problems. Optimization results are obtained by placing the stiffeners inside as well as outside the conical shell. In both these cases, the independent effects of behavior constraints are also studied. The optimum designs are achieved with one of the standard non-linear constrained optimization techniques (Davidon-Fretcher-Powell method with interior penalty function formulation) and few optimal solutions are checked for the satisfaction of Kuhn- Tucker conditions. The researcher concludes that the natural frequency or overall buckling strength is the only behavior constraint considered, it becomes active at optimum point.

***Ding,Y.*[١١]:** treated with finite element analysis and the optimization problem of sandwich construction. The thickness of the faceplates and the core are used as design variables. The hybrid approximation technique in combination with the dual method from mathematical programming is used. Three examples are solved using six-nodal triangular and eight-nodal quadrilateral shell elements. Using six-nodal triangular sandwich is better than using eight-nodal quadrilateral element in similar finite element mesh.

***Ding,Y.*[١٢]:**reviewed the numerical and analytical methods for shape optimization of structures. Several steps in the shape optimization process, such as model description, selection of the objective function and shape variables, representation of boundary shape, finite element mesh generation and refinement, sensitivity analysis and solution methods are reviewed in detail. Eight examples of shape optimization of two-or three- dimensional structures characterizing the state- of – the art in the shape optimization discipline are given.

Marcelin et al [١٣]: concerned with the optimal shape design of thin axisymmetric shells loaded symmetrically or non- symmetrically. The objective is to make a reference stress uniform along a part of the boundary to minimize the stress concentrations. For technological reasons, the boundary of the structure is composed of straight or circular segments defined by input data of master point coordinates and radius values. The design variables are easily deduced from the data. The analysis of the structure is performed by the finite element method using two or three node thin shell elements. Some test examples and an application of the program to the shape optimization of a bottle demonstrate the efficiency of the process. The researcher concludes that the sensitivity analysis has been carried out by a finite difference scheme which is more simple to implement in a finite element code than the usual director or adjoint method because of the very narrow stiffness matrix band-width.

Ostwald et al [١٤]: presented the multi objective optimization of thin-walled sandwich cylindrical shells subjected to axial compression. The optimization problem was given as the first objective function, and the flexibility as second (the flexibility was defined as the susceptibility of shell to deflection). The optimization was carried out using the so- called “Pareto optimality”, in which a set of optimal comprise solutions is generated and then, the best optimal solution is chosen from this set.

Zhiming et al [١٥]: presented the optimal design, nonlinear bending and stability of revolution shallow shells with variable thickness. The problem was investigated by the means of a modified iterative method proposed earlier by the same author. The optimal design of plates and shells, in which the volume is minimized or the critical load of the shell is maximized was investigated. Where the volume of the shell and the arch height of the shell are given, the variable thickness problem was solved. In addition, the paper gave the constraints optimization of circular plates under nonlinear bending.

Chu et al [١٦]: presented a stiffness criterion for ESO, where by stiffness sensitivity number was developed to estimate the change of compliance due to an element removal. It is found in the research presented here in that either the ESO method based on the Von Mises stress criterion or the stiffness criterion can produce almost the same resulting topology.

Visser et al [١٧]: presented the optimal design of buckled and stiffened shear panels in aircraft structures by using a genetic algorithm. The post-buckling behavior of the panels is assessed using the iterative algorithm developed by Grishman. This method requires only a linear finite element analysis, while convergence is typically obtained in as few as ٥ stiffness matrix evaluations. The Grishman algorithm provides for both compressive and shears buckling, and overcomes some of the conservatism inherent in conventional methods of analysis.

***Xie* et al [١٨]:** introduced the modifications and extensions of the ESO algorithm to control the number of cavities to be created in a structure. This modified ESO method, name intelligent cavity criterion (IIC) algorithm takes a different approach to the perimeter method by controlling when a cavity is to be created during the optimization process. This controls the number of cavities and the shape of the cavities due to nature of ESO process. A simple modification of the ESO method allowed the control of a number of cavities producing the less complicated and more manufacturability structures. This will enable ESO to be applied to more practical engineering designs.

Venkataraman [١٩]: presented the design of reusable launch vehicles is driven by the need for minimum weight structures. Preliminary design of reusable launch vehicles required many optimizations to select among competing structural concepts. Analysis and optimization complexities have to be compromised to meet constraints on design cycle time and computational resources. In this work, the use of approximate models and analysis methods implemented using a standard package called PAND٢ software. This software is sued for a trade study to compare weight efficiencies of stiffened panel concepts for a liquid hydrogen tank of a reusable launch vehicles. Optimum weights are obtained for different models. Also in this work, investigate use of response surface approximations for integrating structural response obtained from global analysis in the local optimization of panel. The researcher concludes that the stiffness reduction due to local buckling made stiffener buckling made more critical and increased the bending stresses in the shell.

D.Lyengar [٢٠]: presented the Simultaneous Failure Mode Theory (SFMT) for minimum weight design of structural elements. The approach has been employed to obtain optimum design (minimum strength to weight ratio) of elements like columns, plates, beams, cylinders etc. He was studied the optimization problem for many cases: optimization of the thin walled column elements under axial load, optimum design of wing structures, and minimum cast design of grid floor.

Steven et al [٢١]: showed the equivalence of the von Mises stress criterion and the stiffness sensitivity criterion. Consequently, the criterion of von Mises stress in the classical ESO method is equivalence to the criterion of the compliance minimization or stiffness maximization, which has been exhaustively studied and is well understood in structural optimization. It can be concluded that a stiffness optimization can at the same time yield a strength efficient design from the established stress criterion. From another point of view, the stiffness optimization problem can be solved by directly using the von Mises criterion, ands vice versa.

Ravi S.Bellur [٢٢]: presented a design methodology for the optimization of stiffened plates with frequency and buckling constraints. The basic idea of the methodology is to consider a plate with a fairly dense distribution of stiffeners. Thickness of the plate and stiffeners, and the stiffener width are the design variables. Design variable linking is accomplished by the use of rational spline surfaces. The finite element method is used for the analysis. The plate is modeled using linear Mindlin plate elements and the stiffeners by linear Timoshenko beam elements. Both the plate and the beam elements are shear-locking free by formulation, without requiring any special techniques such as reduced integration. Results for a square stiffened plate with three different stiffener layout patterns and different stiffener density are presented. The best four stiffened configurations which give the lowest mass are chosen and applied to ٢:١ and ٣:١ rectangular plates. It is concluded that the present design methodology gives good results, and that the stiffener pattern and stiffener density play an important role in reducing the mass of a stiffened plate.

Haftka et al [٢٣]: Performed the preliminary optimizations of metallic and laminated composite stiffened shell for reusable launch vehicle liquid hydrogen tank using PANDA٢. Two load cases were chosen: one that was critical for strength requirement(internal proof pressure load) and the other for stability requirement (axial compression load). The optimum weights of shells designed with and without the effect of initial imperfections were compared. For metallic shells, in the absence of imperfections with sufficient design freedom , the panel optimum weight were almost identical presented comparisons of minimum weight designs of hat- stiffened and sandwich cylindrical shells for aluminum and graphite- epoxy materials and discussed the effect of modeling the stiffeners as branched shell elements rather than using a smeared representation. The researcher concludes that the large weight increase of grid stiffened panels can be partly attributed to the use of blonde stiffeners that have small tensional stiffness.

Rigo et al [٢٤]: presented the optimal design of marine structures (ships and naval structures).To perform a rational analysis, the structures modeled using stiffened plate and stiffened cylindrical shell elements in the linear elastic analysis of structures using the so-called “stiffened plate” method to provide a fast and reliable assessment of the stress pattern existing in the stiffened structure. The nonlinear constrained optimization problem was solved by iterative approach. The objective of the design was the minimum cost including raw material, lab our and overhead costs. The design variables were the plate thickness and dimensions of the longitudinal stiffeners. Three types of constraints were distinguished: structural, technological and geometrical.

Lagaros et al [٢٥]: presented the optimum design of stiffened shell structural. Combinatorial optimization methods and more specifically algorithms based on evolution strategies are implemented for the solution of the optimization problem. Three optimization types have been considered: sizing, sizing combined with shape and sizing combined with shape and topology. The problem of shape- topology- sizing optimization of stiffened shell structures with the objective to minimize the weight of the structure under certain behavioral constraints on stresses and displacements is considered. It can be concluded that the beneficial effect of transverse stiffeners on performance of shell structure and the substantial reduction on material volume achieved were demonstrated and assessed quantitatively.

Kasim et al [٢٦]: Studied the optimum design of both multilayer isotropic shell structure and orthotropic shell structure. Due the importance of the application of the shell structure especially in aeronautical fields is considered in this study. In this work the optimum design of multilayer shell is carried out using the Evolutionary structural optimization . In this work both ESO method and the classical optimization methods were studied to verify the efficiency and the generality of using ESO method, which is not existing in any standard package like most classical optimization methods.

Jarmai et al [٢٧]: presented the optimum design of stiffened plates application in airplane wing structures , ship wall and deck structures. Using stiffened plates one can get a light weight and stiffed structure. Several calculation have been developed for stiffed plates. All of them are approximation; the Massonnet and the Gienke techniques. Cost calculation is also important, due to the expensive welding technologies. Two applications are shown: ship deck panel and compressed stiffened plate. It is shown that using optimization, one can reduce the total cost of the structure. In countries where fabrication costs are high the number of stiffeners is small and the thickness is large. In countries where fabrication costs are low the number of stiffeners is large and the thickness is small.

***Lagaros* et al [٢٨]:** investigated the optimum design of stiffened shell structures by using a robust and efficient optimization algorithm where the total weight of the structure is to be minimized subject to behavioral constraints imposed by structural design codes. Evolutionary algorithms and more specifically the evolution strategies (ES) method specially tailored for this type of problems is implemented for the solution of the structural optimization problem. The discretization of the stiffened shell is performed by means of cost-effective and reliable shell and beam elements that incorporate the natural mode concept. Three types of design variables are considered: sizing, shape, and topology.

***Wakeland* et al [٢٩]:** presented the sizing optimizer to optimize topology which was applied to some classical ٢D topology optimization problems. The objective is to minimize the weight. The constraints are the stress or displacement limits. The design variables are the plate thickness. The problem formulation is based on distributed material: plate thickness for continuum structures and cross sectional area for discrete structures, rather than density as is used by many topology optimization methods. At the be ginning finite element meshing is done to discretize the continuum structure. For simple or small problems, each element thickness can be assigned a unique property number as a design variable, and the elements are created to fill the whole reference domain or design space.

***Vigh* [٣٠]:** presented the optimal cross-section geometry of a multi-stiffened web plate girder under dominant bending moment and/or shear. In this presentation, the numerical optimization procedure for symmetric I-girders is in focus. Aim is to achieve the smallest weight with prescribed bending moment and shear.The optimized parameters are the flange width and thickness, web depth and thickness as well as the number of longitudinal stiffeners, their width and height.

Ansola et al [٣١]: presented a modified version of the evolutionary structural optimization procedure for topology optimization of continuum structures subjected to body forces. The approach implemented in this paper is based on the evolutionary procedure. This method has been applied successfully to a variety of problems with different objective functions and multiple constraints, but always considering fixed loads. Body forces depend of the density distribution over the design domain. Therefore, the value and direction of the loading are coupled to the shape of the structure.

٢.٣ Closing Remarks:

Many remarks are extracted from reviewing the literature that concerned with the principles of the correct work these are:

- As shown from the study of O.M.Querin (١٩٩٨)[٢٠]: They concluded that the ESO method, due to its nature, create cavities or holes inside a structural dowain in order to properly distributed the stresses. In order to prevent ESO from generating these cavities a constraint called Nibbling, needs to be in corporate the ESO process . This will enable ESO to be applied to more practical engineering designs. Therefore, this concept will be adopted in the present work.
- Yunliang. D., (١٩٨٦)[١٢]: treated with the finite element analysis and the optimization problem of sand which construction. Thickness of the

faceplates and the Core are used as design variables. Therefore, this element and its formulation will be adopted in the present work.

- It is concluded from the study of G.P.sleven (١٩٩٩)[٢٣] . That the optimum design of general stiffened plate and shell structure by Morphing Evolutionary structural optimization method can be obtained by and reliable method . So any parameter of the stiffening can be set in to the best value with simple procedure . Therefore , an optimization technique will be suggested in the present work based upon this method.
- Referring to the work carried out by R.Ansola (٢٠٠٥)[٣٦] it is clear that using the Morphing method with isotropic plate and shell, the optimization by ESO method is able to produce very good smooth plate thickness variation.

CHAPTER THREE

Finite Element Formulation

٣.١ Introduction

With increasing the use of high speed computer and the emphasis on numerical methods for engineering analysis, the finite element method has developed simultaneously.

The developments of suitable methods, more accurately, the analysis of various engineering structures are needed in order to investigate their behaviour under different loading conditions. Present, the finite element method is a powerful numerical technique, which offers approximate solution to realistic types of structures such as plates and shells. However, the term "finite element" was first used by Clough in ١٩٦٠ (mentioned by Koko, ١٩٨١)**[٤]**. In this method of analysis, a complex region defining a continuum is discretized into simple geometric shapes called "Finite element". The finite element analysis will be used in this work.

In the present study, the ٤-noded isoperimetric quadrilateral elements are used for discrimination of stiffened plates. The derivations of the strain-displacement matrix [B], the elasticity matrix [E] and, as a result, and the stiffened matrix of stiffened plate are presented.

٣.٢ Finite Element Modeling:

Five finite element models are created for the presented study: the first model is prepared for the flat plate, the second and third models are prepared of rectangular-honeycomb and sandwich rectangular –honeycomb, the fourth and fifth models are prepared for the diagonal -honeycomb and sandwich diagonal -honeycomb .

٣.٢.١ Flat plate Finite Element Model

The proposed flat plate finite element model is shown in Fig.. (٣-٣). Linear four- node quadrilateral plane elements are used to build this model. ANSYS finite element package can be created to build and analyze this model, according to the following steps:

١. The domain of the model is assumed to be rectangular surface drawn through the plane of global X and Y axis, their dimensions are L_1 along global X-axis and L_2 along global Y-axis.

٢. The drawn rectangular surface is discretized into ninety six quadrilateral plane elements. Their mesh along global X- coordinate is discretized into twelve elements, each of equally width w_s:

$$w_s=\frac{L_1}{12}.$$

but along global Y- axis, the mesh is discretized into (eight) elements, each of equally width t_s:

$$t_s=\frac{L_2}{8}$$

Figure (٣-٤) shows the location and numbers of the generated nodes, while Fig.(٣-٥) shows the locations and numbers of the generated elements for the same model for L_1=١.٥L_2.

٣.٢.٢ Rectangular-Honeycomb Finite Element Model

The proposed Rectangular-honeycomb finite element model is shown in Fig.(٣-٦). Linear four-node quadrilateral plane elements are used to build this model. The model consists of twenty four rectangular- honeycomb cells equal in dimensions and distributed uniformly along each of global X and Y-axis.

Each cell has (eight) elements bounded by eight nodes. To specify the dimensions of each cell, it's required to know the width, height and thickness of the cell. ANSYS finite element program is used as a mathematical tool in the analysis of this model, with the following steps:

١. Create two rectangular surfaces of width and length along global X and Y-axis respectively of :

$L_١$=٦w_s , and $L_٢$=٤l_s

and separated along global Z-axis by the height of the cell.

٢. The mesh density is specified on each surface such that they are discredited into four segments along the two directions. Each segment along global X-axis is equal to (w_s),while along Y-axis is equal to (l_s). Then node generation is done on each surface according to specific density.

٣. The elements are connected thought the nodes that are located on any of the surfaces and that corresponding on the other surface, along the plane of global X and Z-axis and that of global Y and Z-axis.

Figure (٣-٧) shows the location and numbers of the generated nodes, while Fig.(٣-٨) shows the elements locations and numbers of this model.

٣.٢.٣ Diagonal-Honeycomb Finite Element Model

The diagonal-honeycomb finite element model is shown in Fig. (٣-١٠). Linear four-node quadrilateral plane elements are used to build this model. The model consists of seven rhombic-honeycomb cells equal in dimensions and distributed uniformly along global X and Y-axis. Each cell has eight elements bounded by sixteen nodes. To specify the dimensions of each cell, it's required to know the quadrilateral-side length and the height of the cell. ANSYS finite element program is used as a mathematical tool in the analysis of this model, with the following steps:

١. Create two rectangular surfaces of width and length along global X and Y-axis respectively of :

   $L_{١}$=٢l_s , and $L_{٢}$=٣l_s

   and separated along global Z-axis by the height of the cell.

٢. The mesh density is specified on each surface such that they are discredited into four segments along the two directions. Each segment along global X-axis is equal to (w_s), while along Y-axis is equal to (l_s). Then node generation is done on each surface according to specify density.

Figure (٣-١١) shows the location and numbers of the generated nodes, while Fig.(٣-١٢) shows the elements locations and numbers of this model.

٣.٣ Coordinates Systems

There are three types of coordinates systems used in above finite element models, these are:

١- the global coordinates system.

٢- the nodal coordinates system.

٣- The element coordinates system.

٣.٣.١ Global Coordinate Svstem

The global coordinates system, which is used in the above model:, is a fixed Cartesian coordinates system. All surfaces, nodes and elements of the models are drawn with respect to the global coordinates system. In addition, the direction of applied loads and the constraint degrees of freedom (boundary conditions) are defined along these coordinates system. The analysis result included linear and angular displacements of the nodes under action of applied static loads, the directions of these displacements are given along the global coordinates axis.

٣.٣.٢ Nodal Coordinates System

To describe the direction of the element degrees of freedom, a nodal coordinates system created on each node must be used. The nodal coordinates system is located at each node as shown in Fig.(٣-١) such that the origin is located at the corresponding node position. Nodal x-axis is tangent to the curve joining this node and the subsequent nodes. Nodal y-axis is tangent to the element surface, its direction perpendicular to that of nodal x-axis. The nodal z-axis is perpendicular to the x-y plane, its direction is specified according to the orthogonal right- handed rule.

٣.٣.٣ Element (local) Coordinates Svstem

The element coordinates system is located at each element as shown in Fig. (٣-١) . This coordinates system is created on each element such that,its origin located at the element areas center. The element s-axis is tangent to the element surface at coordinates origin along the direction of the line joining the first and second nodes through element connection in the direction from first node to the second node. The element t-axis is located on the element plane such that, its direction perpendicular to that of element x-axis on the side where the third node located. The element r-axis is perpendicular to the element plane, towards the orientation vector of the element connection loop as shown in Fig. (٣.٢) . The element coordinates system is used to describe the direction of the applied loads and direction of the stresses and strains that induced in the elements.

٣-٤ Stress-Strain Relation-Ship

The stress- strain relations in coordinates aligned with principle material directions are given by:**[٣].**

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} c_{11} & c_{12} & 0 & 0 & 0 & 0 \\ c_{21} & c_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & c_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & c_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & c_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \quad ..(٣\text{- }١)$$

From the usual thin plate assumption, the normal stress σ_z is assumed small enough to be neglected and the corresponding ε_z is eliminated. **[Ding, ١٩٨٦].** Therefore, the equation (٣-١) becomes:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} c_{11} & c_{12} & 0 & 0 & 0 \\ c_{21} & c_{22} & 0 & 0 & 0 \\ 0 & 0 & c_{44} & 0 & 0 \\ 0 & 0 & 0 & c_{55} & 0 \\ 0 & 0 & 0 & 0 & c_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} \quad ..(٣\text{-}٢)$$

or

$$\{\sigma\} = [E]\{\varepsilon\}$$

where:

$c_{١١}=c_{٢٢}=(١-\nu)/A$ $c_{١٢}=c_{٢١}=(\nu)/A$ $c_{٤٤}=G$

$c_{٥٥}=fG$ $c_{٦٦}=fG$

$A=\frac{(1-\nu)(1-2\nu)}{E}$, $G=\frac{E}{2(1+\nu)}$

Where f is the shear factor for homogeneous plate should be given a value of ١.٢ in order to account for the fact that the transverse shearing stresses produce too little strain energy **[٤٧].**

٣.٥ Element Parameters

All above finite element models have been created using linear four-node quadrilateral plane elements. This type of elements is used for plate and shell structures for both membrane and flexure load conditions. In this section, the parameters that are concerned with the selected element are discussed.

These parameters are basically included; the element property parameters include the material properties and the thickness of the element at each node.

For the rectangular- honeycomb finite element models, the material properties for all elements are specified as isotropic material. The ration of thickness value to the smallest element dimension must be equal or less than (٠.١) in order to maintain the element to be thin.

The element degrees of freedom are assigned at each node along the element coordinate system. Figure (٣-٢) shows the element degrees of freedom of any point located on the element is function of that of all element nodes ,as:-

$$\begin{Bmatrix} u(\xi,\eta) \\ v(\xi,\eta) \\ w(\xi,\eta) \end{Bmatrix} = \sum_{i=1}^{4} N_i(\xi,\eta) \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} + z \sum_{i=1}^{4} N_i(\xi,\eta) \begin{Bmatrix} \theta_{xi} \\ \theta_{yi} \\ 0 \end{Bmatrix} \qquad ..(٣\text{-}٣)$$

Where: - z=- $\frac{t}{2} \rightarrow \frac{t}{2}$

$$\left.\begin{aligned} N_1(\xi,\eta) &= \frac{1}{4}(1-\xi)(1-\eta). \\ N_2(\xi,\eta) &= \frac{1}{4}(1+\xi)(1-\eta). \\ N_3(\xi,\eta) &= \frac{1}{4}(1+\xi)(1+\eta). \\ N_4(\xi,\eta) &= \frac{1}{4}(1-\xi)(1+\eta). \end{aligned}\right\} \qquad ..(٣\text{-}٤)$$

where:-

N_i=Shape functions.

u_i , v_i , w_i= global nodal displacements .

θ_{xi} , θ_{yi} = global nodal notations .

z = nodal thickness.

It is obvious that each node has five degrees of freedom, and then the element is of twenty degrees of freedom. Not all but some of the element degrees of freedom are considered at each of the finite element models, depending upon the function (boundary conditions) of that model.

٣.٦ Derivation of Strain – Displacement Matrix:-

The geometrical nonlinearity is not considered in the present work, and hence the engineering components of strain can be expressed in terms of first partial derivatives of the displacement components. Therefore, the linear strain – displacement matrix,[B] at any point an element, and for a seven degrees of freedom per nod can be written as:

$$\{\varepsilon\}=\begin{Bmatrix}\varepsilon_{xm}\\ \varepsilon_{ym}\\ \gamma_{xym}\\ \varepsilon_{xb}\\ \varepsilon_{yb}\\ \gamma_{xyb}\\ \gamma_{xz}\\ \gamma_{yz}\end{Bmatrix}=\begin{bmatrix}\frac{\partial u_o}{\partial x}\\ \frac{\partial v_o}{\partial y}\\ \frac{\partial u_o}{\partial y}+\frac{\partial v_o}{\partial x}\\ \frac{\partial \theta_x}{\partial x}\\ \frac{\partial \theta_y}{\partial y}\\ \frac{\partial \theta_x}{\partial y}+\frac{\partial \theta_y}{\partial x}\\ \theta_x+\frac{\partial w_o}{\partial x}\\ \theta_y+\frac{\partial w_o}{\partial y}\end{bmatrix}=\begin{bmatrix}\frac{\partial N_i}{\partial x} & 0 & 0 & 0 & 0\\ 0 & \frac{\partial N_i}{\partial y} & 0 & 0 & 0\\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 & 0 & 0\\ 0 & 0 & 0 & \frac{\partial N_i}{\partial x} & 0\\ 0 & 0 & 0 & 0 & \frac{\partial N_i}{\partial y}\\ 0 & 0 & 0 & \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x}\\ 0 & 0 & \frac{\partial N_i}{\partial x} & N_i & 0\\ 0 & 0 & \frac{\partial N_i}{\partial y} & 0 & N_i\end{bmatrix}\begin{Bmatrix}u_o i\\ v_o i\\ w_o i\\ \theta_x i\\ \theta_y i\end{Bmatrix} \quad ....(3-5)$$

Since the shape functions N are functions of the local coordinates rather than Cartesian coordinates, a relationship needs to be established between the derivatives in the two coordinates systems. By using the chain rule, the partial differential relation can be expressed in matrix form as:

$$\begin{bmatrix}\frac{\partial N_i}{\partial \xi}\\ \frac{\partial N_i}{\partial \eta}\end{bmatrix}=\begin{bmatrix}\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi}\\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta}\end{bmatrix}\begin{bmatrix}\frac{\partial N_i}{\partial x}\\ \frac{\partial N_i}{\partial y}\end{bmatrix} \quad ...(٣-٦)$$

where [J] is the Jacobian matrix and the elements of this matrix can be obtained by differentiating the following equations:

$$x(\xi,\eta)=\sum_{i=1}^{4} N_i(\xi,\eta)x_i \quad ...(٣-٧)$$

$$y(\xi,\eta)=\sum_{i=1}^{4} N_i(\xi,\eta)y_i \quad ...(٣-٨)$$

Hence, the Jacobian matrix can be expressed as:

$$[J]=\begin{bmatrix} \sum_{i=1}^{4}\frac{\partial N_i}{\partial \xi}x_i & \sum_{i=1}^{4}\frac{\partial N_i}{\partial \xi}y_i \\ \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}x_i & \sum_{i=1}^{4}\frac{\partial N_i}{\partial \eta}y_i \end{bmatrix}$$ ...(٣-٩)

then, the derivatives of the shape function with respect to Cartesian coordinates can be given as:

$$\begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \end{bmatrix}=[J]^{-1}\begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \end{bmatrix}$$ ...(٣-١٠)

where $[J]^{-١}$ is the inverse of Jacobian matrix given as:

$$[J]^{-1}=\begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix}$$ ...(٣.١١)

٣.٧ Derivation of Element Stiffness - Matrix

The basic concept of the finite elements method is to discrete the continuum into arbitrary numbers of small element connected together at their common nodes. The strain – displacement matrix, [B], as shown previously is given by

$$\{\varepsilon\}=[B]\{u_i\}$$ ...(٣-١٢)

The total solution domain is discredited into a number of element (NE) [sub- domain] such that:

$$\pi(u)=\sum_{e=1}^{NE}\pi^e(u)$$ ...(٣-١٣)

Where, π and π^e are the potential energies of the total solution domain and the sub-domain, respectively. The potential energy for an element, e, can be expressed in terms of the internal strain energy, SE, and external work done, **WF**, which can be written as:-

$$\pi^e(u) = SE - WF \qquad ...(٣.١٤)$$

In which (u) is the vector of nodal degrees of freedom of an element. The internal strain energy of a linearly elastic body is given by:

$$SE = \frac{1}{2}\int_A \varepsilon^T \sigma dA \qquad ...(٣\text{-}١٥)$$

Where the integration is over the area of the element.

By substitution ($\sigma = E\varepsilon$) and ($\varepsilon = [B].u_i$) in the equation above, then :-

$$SE = \frac{1}{2}\{u\}^{eT} \int_A [B]^T [E][B] dA \{u\} \qquad ...(٣\text{-}١٦)$$

The external work done by uniformly distributed load is given by:

$$WF = \int_A \{u\}[p]\, dA \qquad ...(٣\text{-}١٧)$$

But, the displacement vector {u} can also be defined as:-

$$\{u\} = [N]\{u_i\}^e \qquad ...(٣\text{-}١٨)$$

And by substation equation (٣-١٧) in equation (٣-١٨) one get:

$$WF = \int_A \{u\}^{eT} [N]^T [p]\, dA \qquad ...(٣\text{-}١٩)$$

$$\pi^e(u) = \frac{1}{2}\{u\}^{eT} \int_A [B]^T [E][B]\, dA \{u\}^e - \int_A \{u\}^{eT} [p]\, dA \quad ...(٣-٢٠)$$

To obtain the Equilibrium state of the element, the potential energy must be minimized with respect to nodal displacements as follows:

$$\{\frac{\partial \pi}{\partial u^e}\} = \{0\} \quad ...(٣-٢١)$$

By substitution of equation (٣-٢٠) in the equation (٣-٢١) and doing the partial differentiation, one gets:-

$$\left\{\frac{\partial \pi}{\partial a^e}\right\} = \int_A [B]^T [E][B] dA \{u\}^e - \int_A [N]^T [P] dA = \{0\} \quad ...(٣-٢٢)$$

or

$$[k]^e \{u\}^e - [F]^e = \{0\} \quad ...(٣-٢٣)$$

where,

$$[k]^e = \int_A [B]^T [E][B]\, dA = \int_{-1}^{1}\int_{-1}^{1} [B]^T [E][B] |J| d\xi d\eta \quad ...(٣-٢٤)$$

$$[F]^e = \int_A [B]^T [p] dA = \int_{-1}^{1}\int_{-1}^{1} [N]^T [E][p] |J|\, d\xi d\eta \quad ...(٣-٢٥)$$

In which

$[k]^e$: is the element stiffness matrix.

$[F]^e$: is the element external applied force vector.

$|J|$: is the determinate of the Jacobian matrix

٣.٨ Boundary Conditions:

The specify of boundary conditions includes: the assignment of the constraint and free state of the degrees of freedom at each node in the finite element models. The state of the degree of freedom if it is constrained or free depending upon the model itself. Thus for each model, there are different sets of boundary conditions created through their corresponding computer programms for each function.

From the first model to the fifth model , the model is fixed from the side joining nodes of numbers(١-٢٩ step ٧) and that joining nodes of numbers (٣٦-٦٤ step ٧), thus all these nodes are constraint in all their degrees of freedom

٣.٩ Static Analysis

Static analysis is achieved on each of the finite element models for each function with their corresponding boundary conditions and load sets. Static analysis solution has been included the calculation of the effects of the applied static distributed loads on each model for each function with the corresponding boundary conditions. These effects included displacements, strains, and stresses that are induced in the structure due to the applied loads. The static analysis is governed by the following equilibrium equations (in matrix notation):

$$[K].\{u\}=\{F\} \qquad ..(٣-٢٦)$$

Where: $[K]=\sum_{e=1}^{N} [K]^{e}$:the assembled stiffness matrix.

ANSYS package solve the above equilibrium equations to obtain the following results:

- Displacements of each node along their free degrees of freedom.
- Strains, and stresses at each element along element coordinate axis.
- Von Misses stresses and the maximum shear stresses[appendix-B]

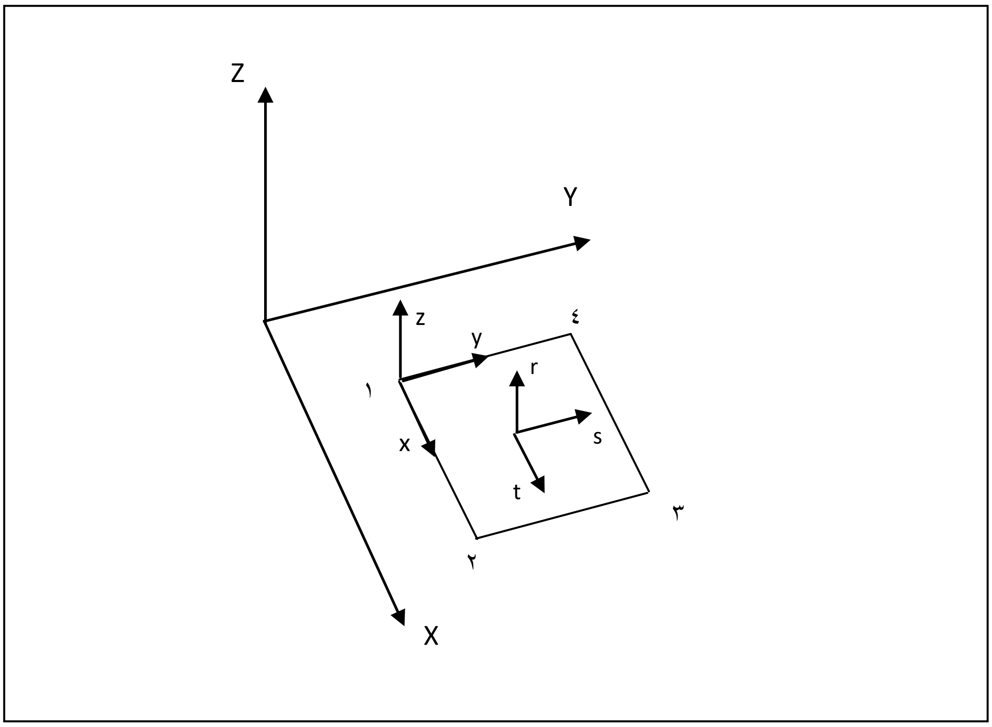
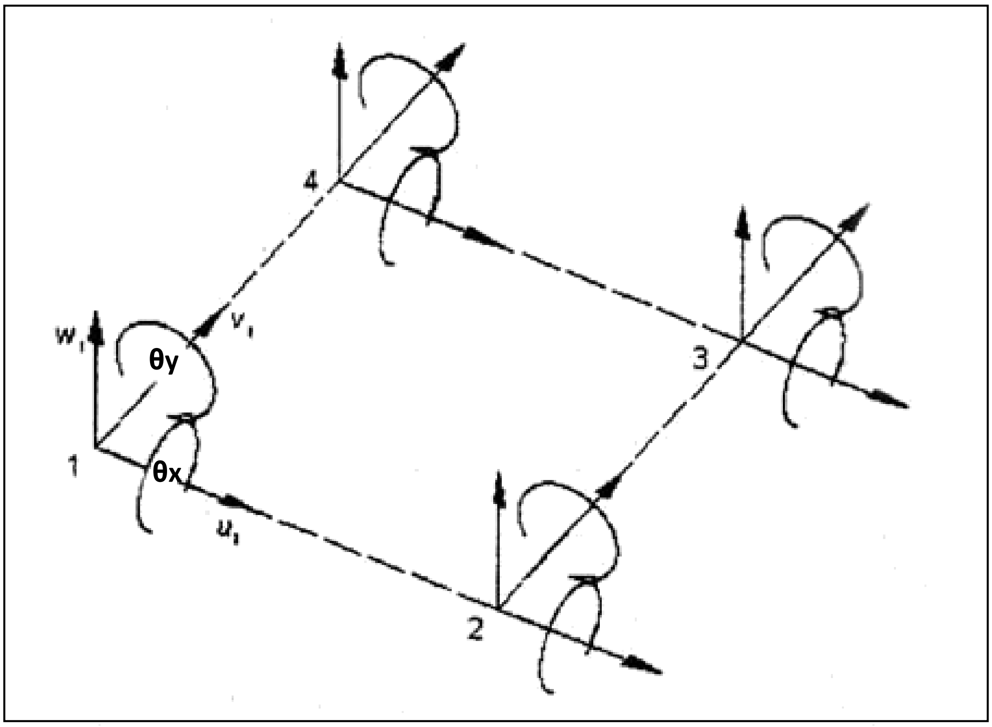
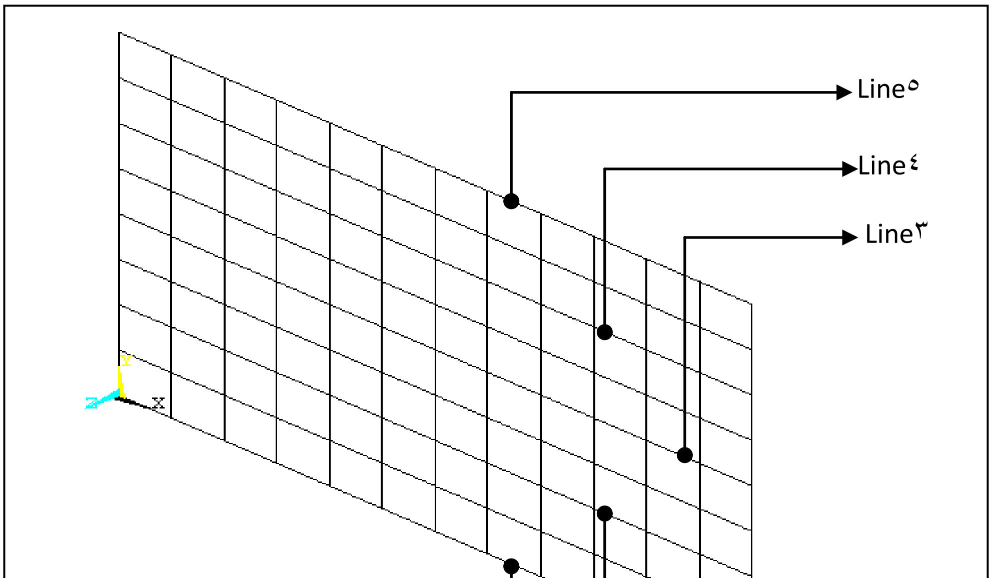
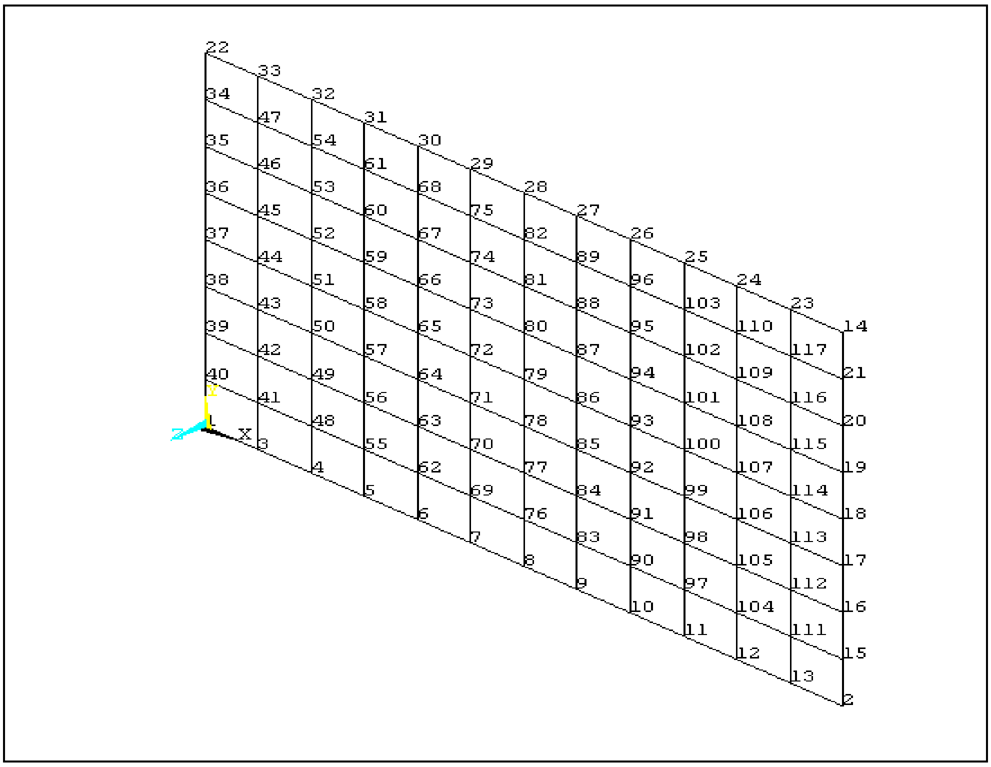


Fig.(٣-١):The element coordinates system

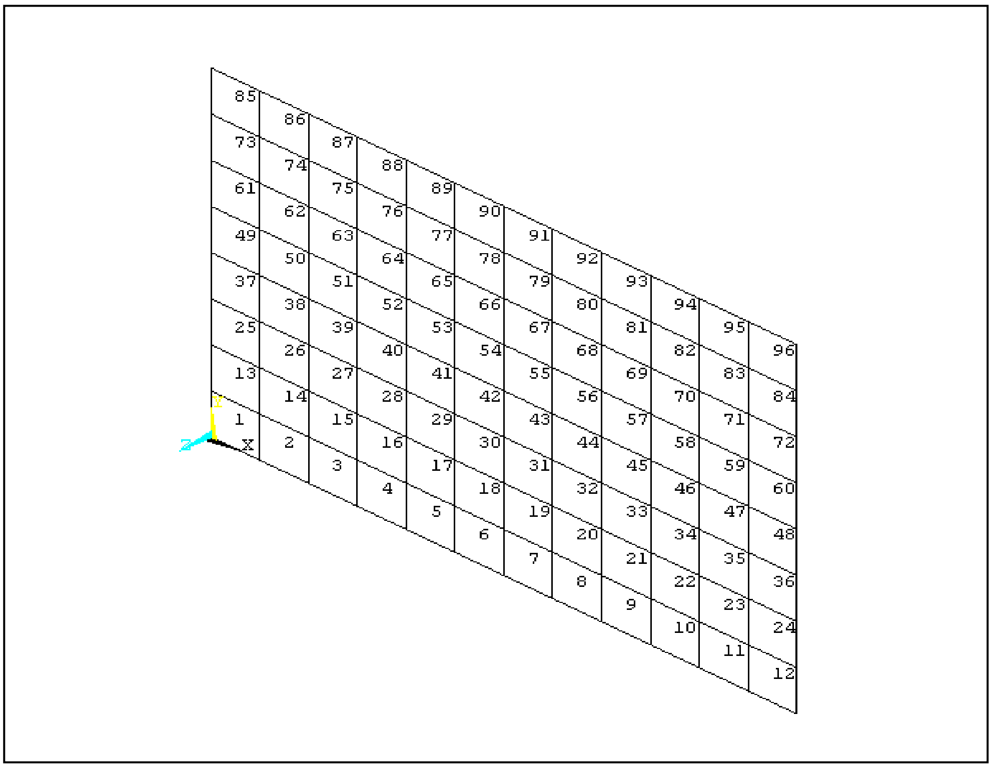


Fig(٣-٢): The element degrees of freedom

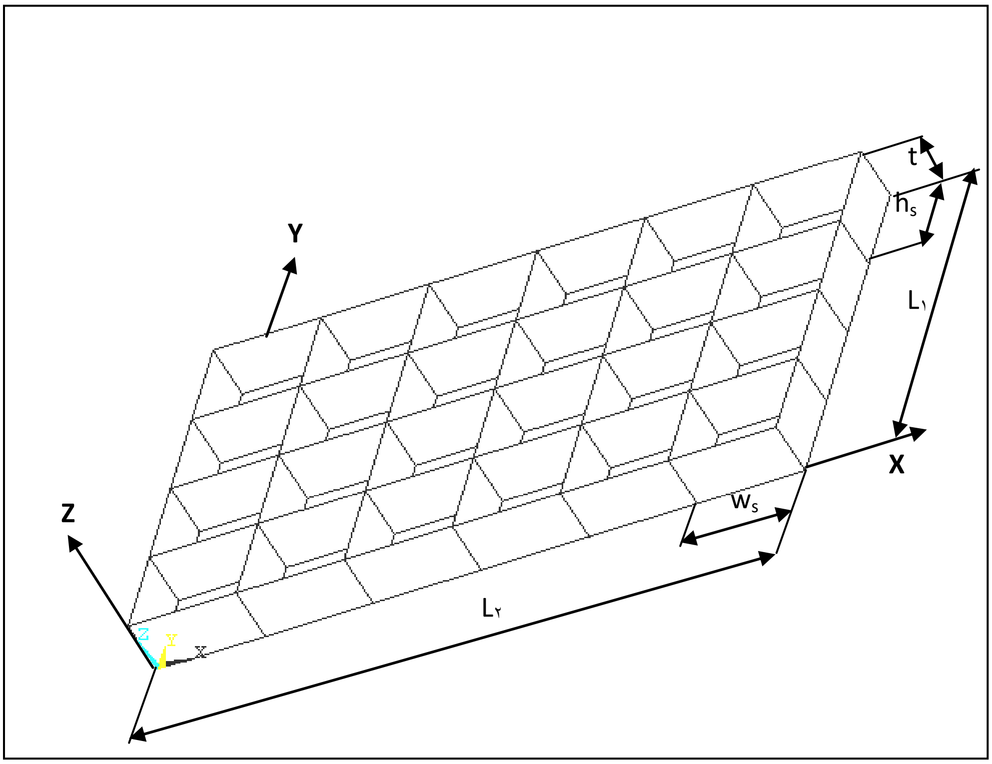




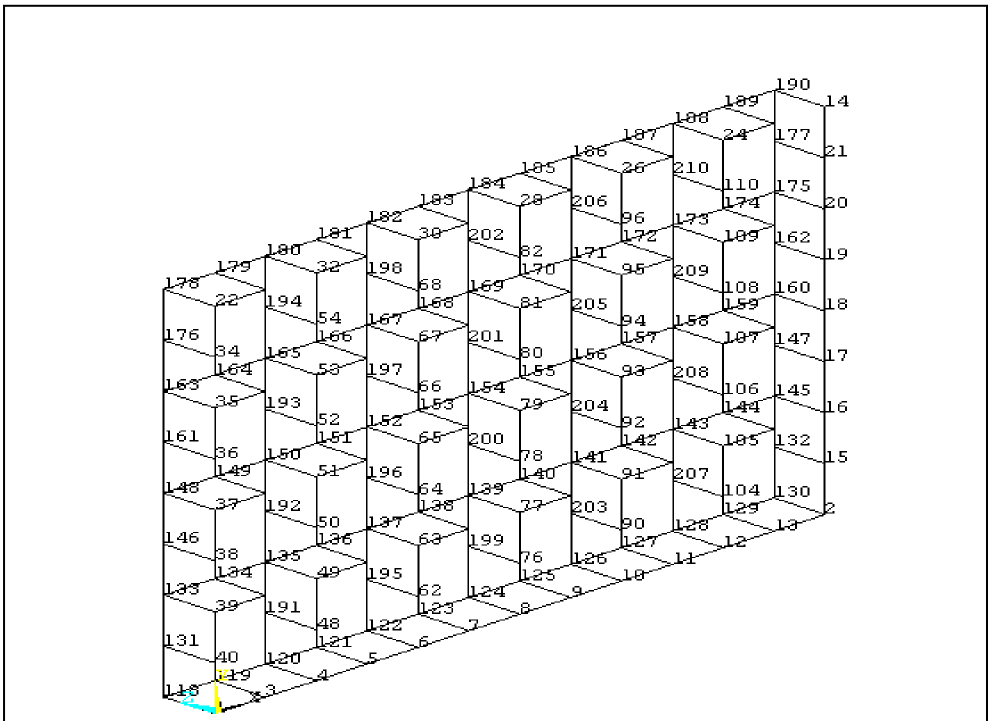
Fig(٣-٤):The location and number

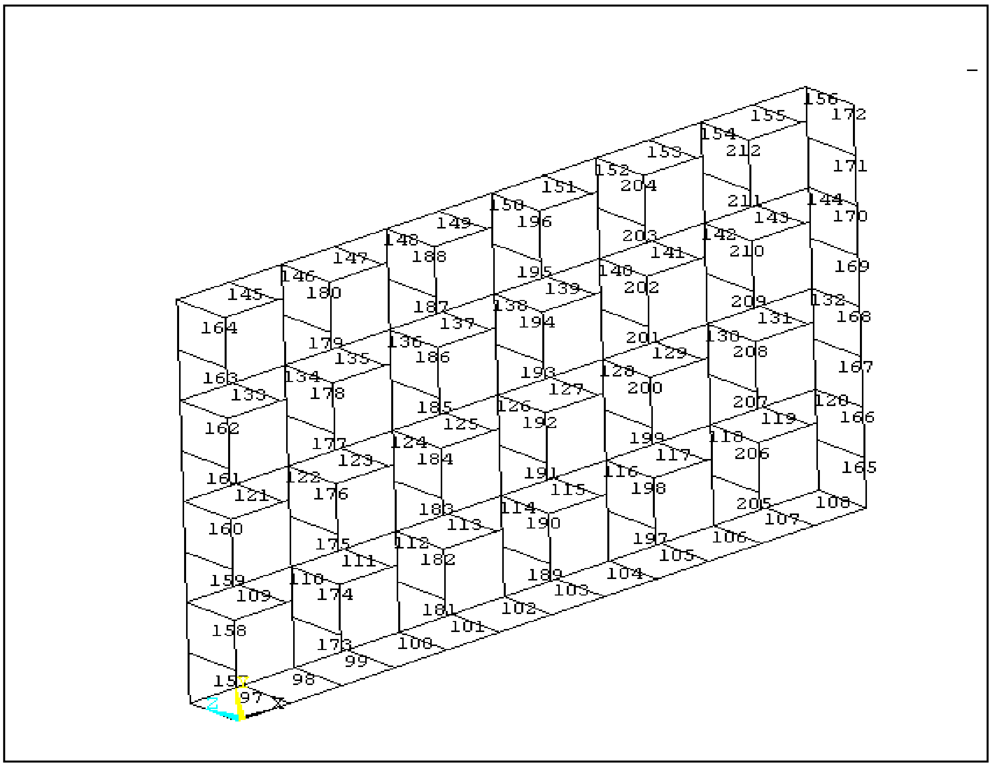


Fig(٣-٥):The location and number

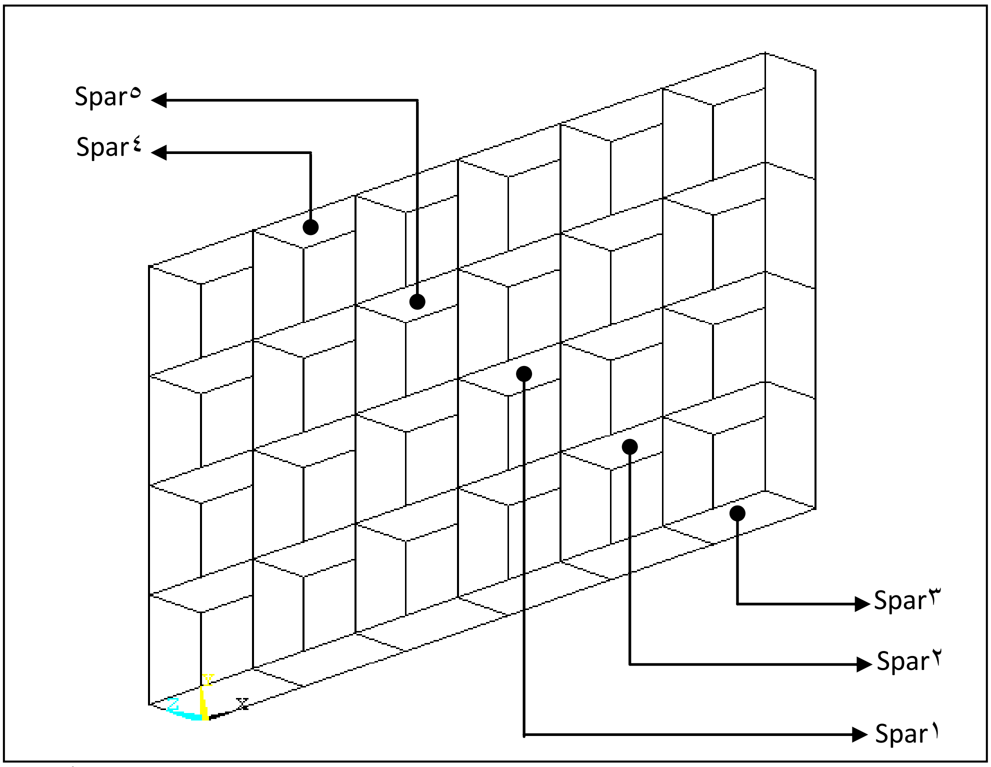


Fig(٣-٦): Rectangular-honeycomb finite element model

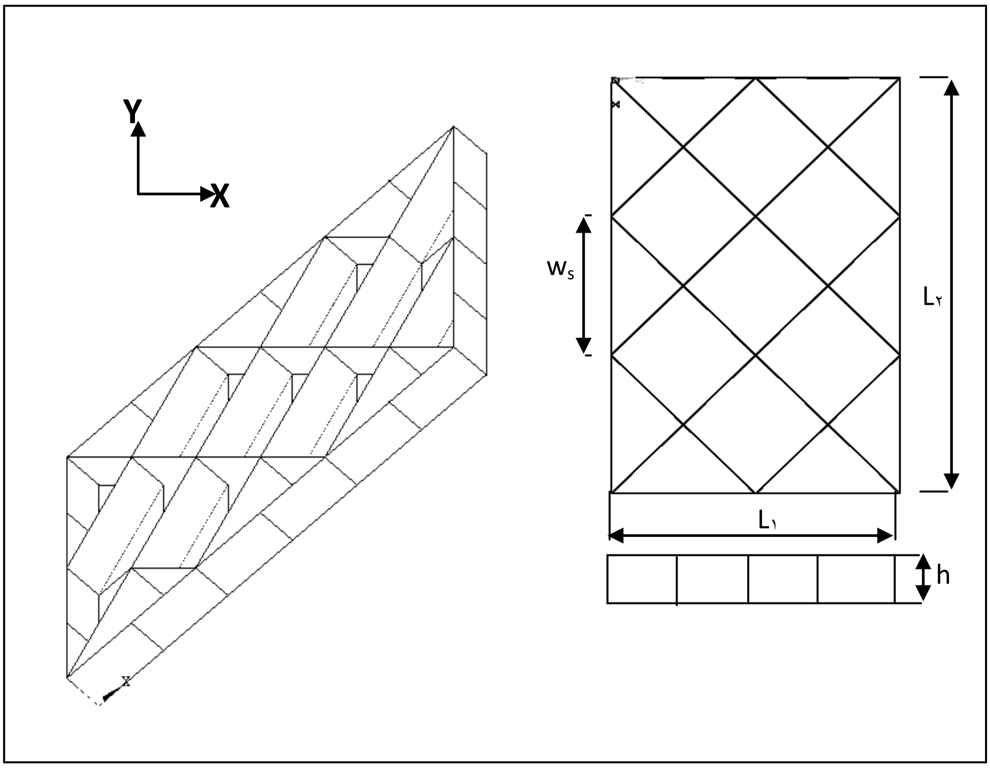




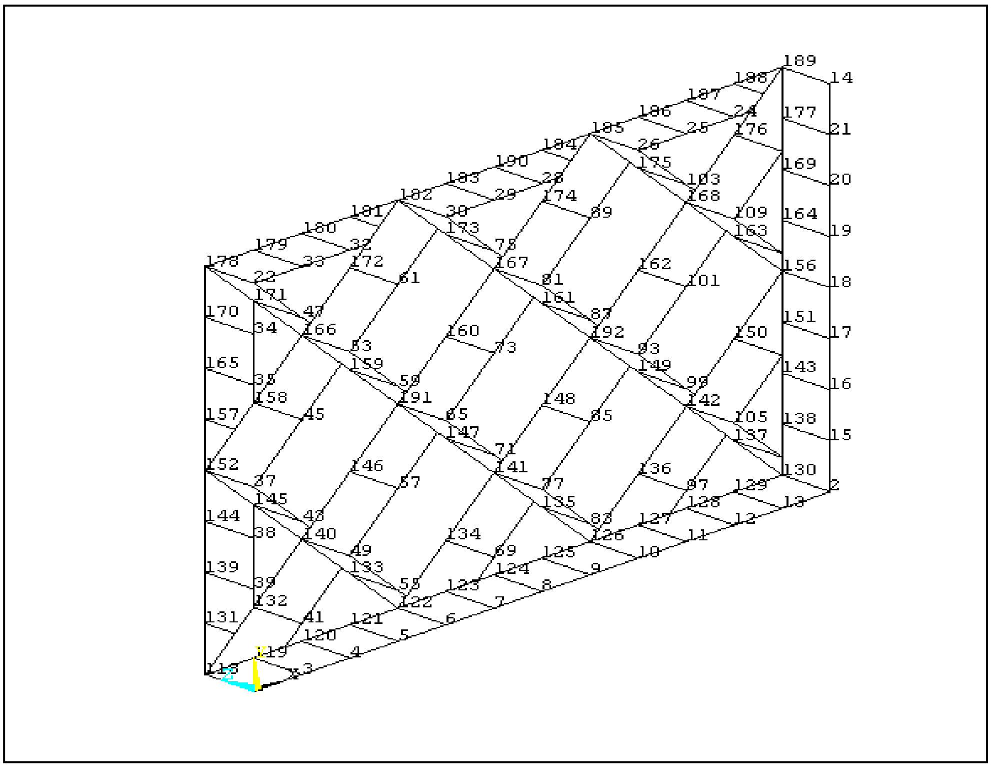
Fig(٣-٨):Locations and numbers of



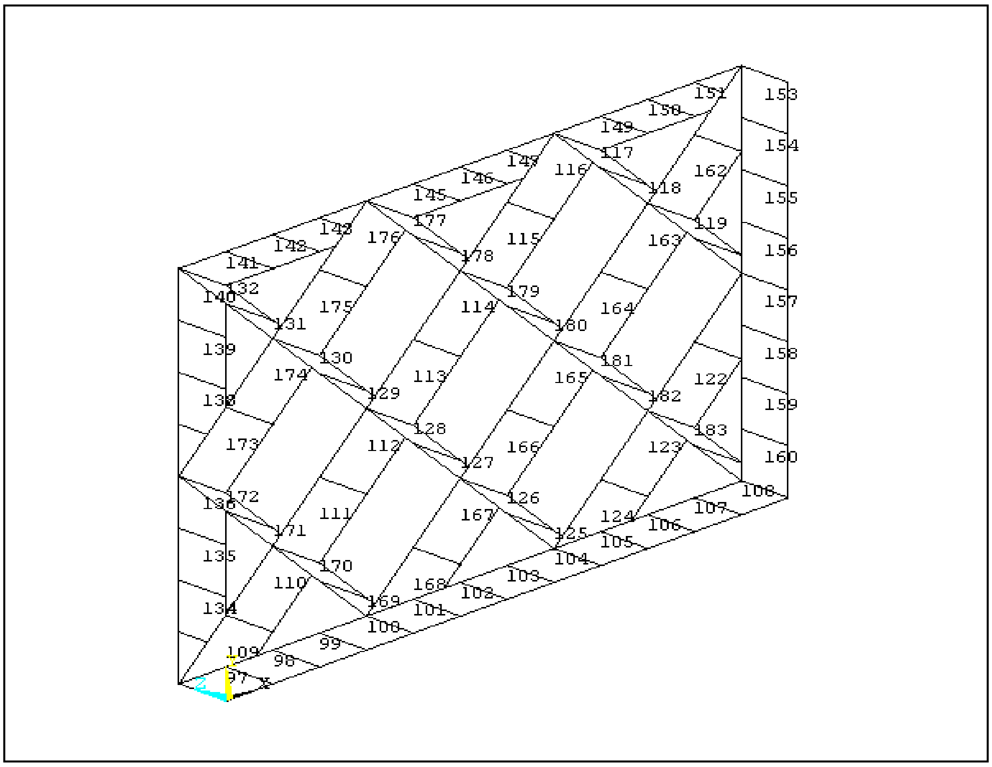
Fig(٣-٩):Presents Spars indices



Fig(٣-١٠): Rhombic -honeycomb finite element mode:



Fig(٣-١١):The location and numbers of



Fig(٣-١٢):The location and numbers of