مجلة جامعة بني وليد للعلوم الإنسانية والتطبيقية
تصدر عن جامعة بني وليد – ليبيا
Website: https://jhas-bwu.com/index.php/bwjhas/index
المجلد التاسع، العدد الثاني 2024

إيجاد الاجهاد الكلى والنصف قطري لأحجام مختلفة من المحددات باستخدام ومقارنة الحسابات الرياضية *I-DEAS*

أ. سعد صالحين محمد خلف الله
قسم الأشعة ، كلية التقنية الطبية ،جامعة بنى وليد ليبيا
Saadalbabaa@bwu.edu.ly

Using I-DEAS Program and equation mathematic for the radial and hoop stress for different meshes densities

Saad Salhin Kalfalla
Department of Radiology, College of Medical Technology, Bani Waleed University, Libya

تاريخ الاستلام:01-05-2024 تاريخ القبول: 18-05-2024 تاريخ النشر : 05-06-2024

الملخص:

لقد تم في هذا البحث مناقشة بعض المشاكل المتعلقة بدراسة المقارنة في تحليل العناصر المحددة في مرحلة التصميم الهندسي للصناعات الميكانيكية في فترة زمنية سريعة من اجل الحصول على نتيجة عالية الدقة وذللك باستخدام طريقتين الاولى باستخدام المعادلات الرياضية والطريقة الثانية باستخدام برنامج (**IDEAS SOFTWARE**) على جدار الاسطوانة الهندسية ، حيث قمنا باتخاذ الخطوات المناسبة لقياس الاجهاد القطري والنصف القطر بأحجام مختلفة من الشبكات **cylinder plate** لذا فقد احتوى البحث على لمحة عامة عن طريقة تحليل **FEA** وبعض الحلول للمشاكل الهندسية الصادرة عن **I-DEAS** ومقارنة أحجام الشبكات المختلفة (5*8، 16*16، 32*32، 64*64، 128 *128). قيد الإزاحة لكلا الجانبين الداخلي والخارجي للاسطوانة . ولقد تم تحديد القوة المطبقة بمقار 10 نيوتن/مم^2 لإيجاد نوع الانفعال الخطي للمادة لإيجاد نتيجة مبدأ الحد الأقصى والأدنى لحجم ا لشبكة للعنصر الأول و باستخدام نفس الخطوات لكل حجم شبكة من الاحجام الاخرى. وعند القيام بالطريقة الثانية وهى الحساب بالمعادلات الرياضية وجدنا ان كلما زاد حجم الشبكة في برنامج I-DEAS كلما اقترب من التطابق في النتيجة مع الحسابات بالمعادلات الرياضية.

الكلمات الدالة: دراسة، الإجهاد، النصفي، القطري، الأسطوانة، IDEAS

Abstract

During this research, there are two methods have used , they are ;modern IDEAS software and hand calculation for solving the comparisons problem in define elements analysis in a period of time, because of getting too high perfect result. And also it has taken the steps to measure radial and hoop stress for different mesh size to beat problem of mechanics in wall cylinder. So the research has contained overview about FEA analysis way and some of solution for problems of engineering that is released on I-DEAS and compare the different of mesh size (5*8, 16*16, 32*32, 64*64 and 128*128) Displacement constraint for both side. In this research deals with the force by 10 N/mm^2 and show the kind of solution linear strain , show the material to find the result of maximum , minimum principle for first element mesh using the same steps for each mesh size. Through work has taken some steps by calculating the hoop stress to the both inner Radius stress that can solve by steps which are performed of suddenly failure of mechanical materials in stout cylinder designer.

Keywords: Study, stress, radius, cylinder, IDEAS

INTRODUCTION

The FEA system consists of a computer model of a material or design that has concerted and analyzed for specific results. It is used in new product design, and existing product improvement. From here, a company will have ability to verify a proposed design and able to perform for the specifications of the client before to manufacturing or building. By using the modifying an existing product or structure for qualifying the product or structure a new service case. When structural failure, here FEA can be used for helping determine the design modifications to do the new condition. In general there are two kinds of analysis which are used in industry;2D and 3D modeling There is a different between 2D and 3D to get the results, so 2D modeling keeps simplicity and allows the analysis to be done a relatively normal computer, it tends to get less accurate results. However the 3D modeling are has ability to produce more accurate results with sacrificing the ability to take on all except the fastest computers effectively.

Within each of these modeling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and do not take into account plastic deformation generally . Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

The stresses generated in the objects made Designers given the most importance during design the objects. This is led them to devise modern methods to help them to the discovery early weakness in the objects. Man-made objects have a purpose and a nominal specification. Durability is an important aspect of the specification of built or manufactured items. Designers and engineers use materials data and expected load levels to create suitably sized components through the use of stress analysis. Stress is most fundamentally expressed as load divided by area. It expresses the severity of loading experienced by material in equipment (hence, higher loads can be sustained by larger cross-sections of the same material). A variety of reasons, however, mean that stress is never uniform in engineering components. Stress distributions are frequently complex. The methods of stress analysis have been developed to give engineers practical methods for assessing the level of stresses.

Computer based numerical stress analysis methods (e.g. finite element analysis or FEA) have permitted the complex distributions of stress in engineering components to be more routinely calculated. These allow linear elastic and non–linear (e.g. elastic–plastic) stress analyses to be performed for static and dynamic loads. The computer–based engineering office is now integrating the processes of component drawing and component stressing through the use of Computer Aided Design (CAD) systems. Information transfer should therefore be more straightforward. However, advanced software is needed to interpret and process the data. Further development of the software and the file format protocols is needed. Existing engineering components can be assessed using experimental stress analysis. The results will then be applicable to the real material characteristics of the component and the correct in–service loads. Imanijed& G. Subhash[3] developed a generalized solution for small plastic deformation of thick– walled cylinders subjected to internal pressure and proportional loading.

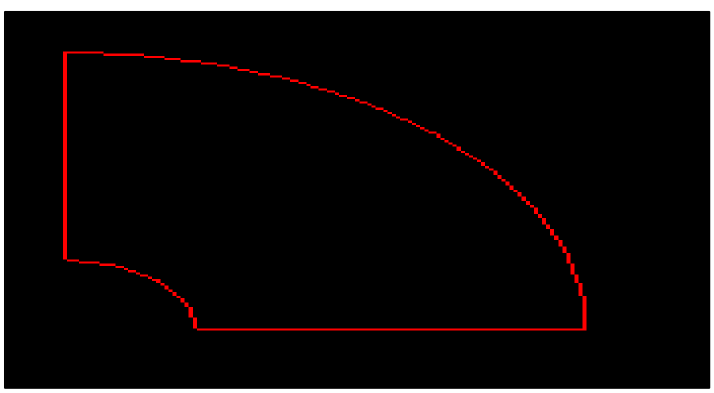
OBJECTIVE OF RESEARCH:

The aim of this study is spotlight on:

discussed the finite element analysis with deferent mesh size ,the result of hoop ad radial stress by I–DEAS and hand calculation by using the Equation and comparing between all samples to getting too high perfect result.

Methodology Of Research

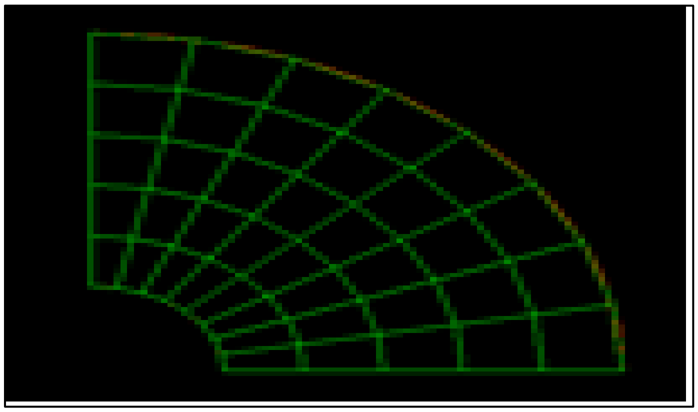
Research uses FEA analysis by IDEASE software to calculate theses stresses by using 2D different mesh sizes . First of all, we prepare the I–DEAS software to draw the whole part, (simulation and model solution), and draw the part by dimension, by thickness Set unites to mm (Newton) and Draw two circles one with radius (20 mm) and the other one with (5 mm). we Use polylines to draw two lines from the center to the end of small circle and to the end of big circle. chose boundary definition using surface by boundary , the sample of pipe will appear as in the figure 1 below.



Figure(1) shows the sample of pip

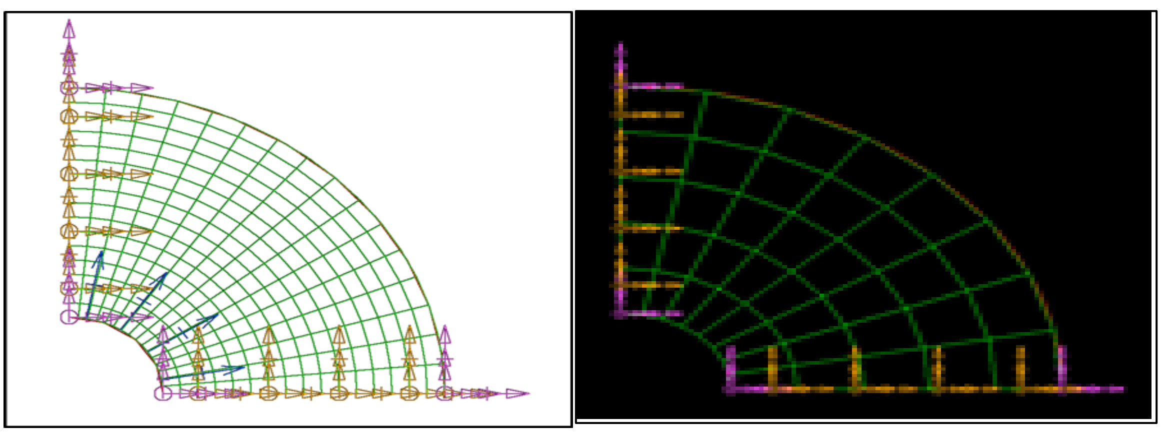
1- Sample (5*8)

After that we Create an Model on the sample of pipe by using FE , define the surface to define shell mesh define elements sides then insert number of nodes that you wish then press mesh then press keep mesh, after complete mesh the surface the sample of pipe will appear as in the figure 2 below.



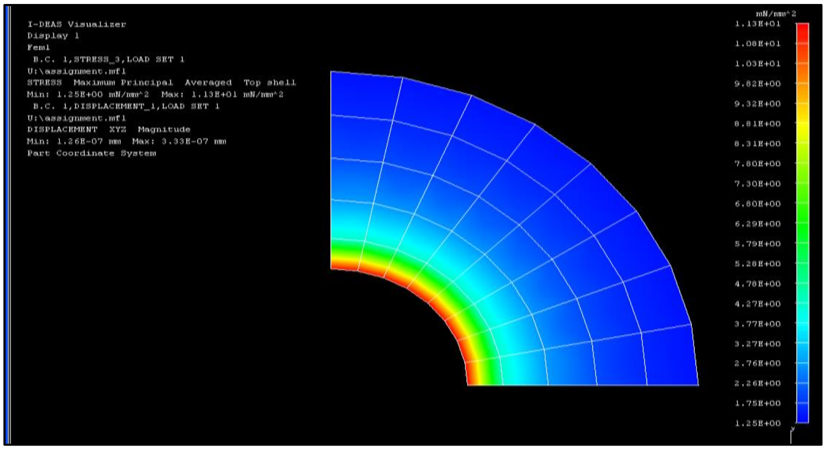
Figure(2) shows how to use displacement restraint

After complete the fully fixed restraint the sample of pipe will appear as in the figure 3 below.



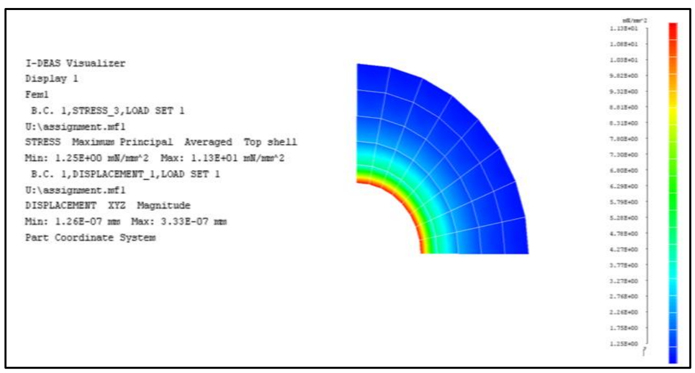
Figure(3) shows the fully fixed restraint after create

create the force by which is axisymmetric intensity force/area .Created a ‘Boundary Condition Set’ .click boundary condition sets then click new set then put restraint set ON then from load sets click in the load set 1then press OK. After that has done ‘Solution Set’ .the window will appear which include solution set as shown in figure 4

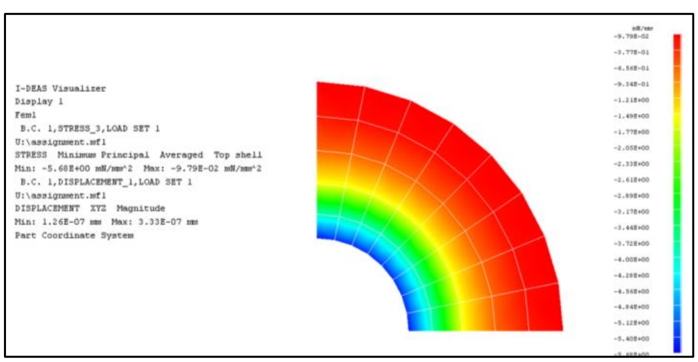


Figure(4) shows the sample of 5*8 after created.

display the stress select maximum principal and minimum principal as shown in figures 5 ,6



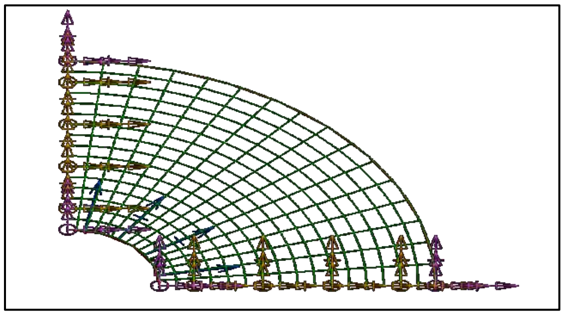
Figure(5) shows the Maximum Principal stress of (5*8)



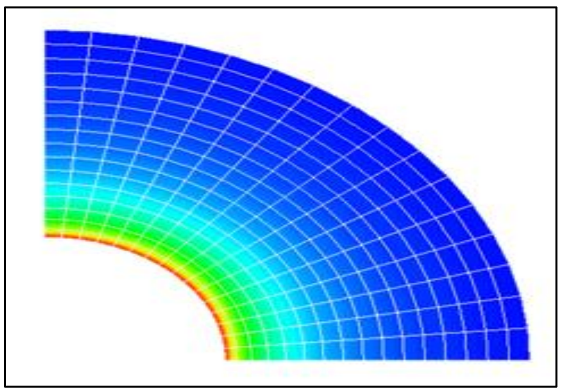
Figure(6) shows the Minimum Principal stress of (5*8)

2.Sample (16*16)

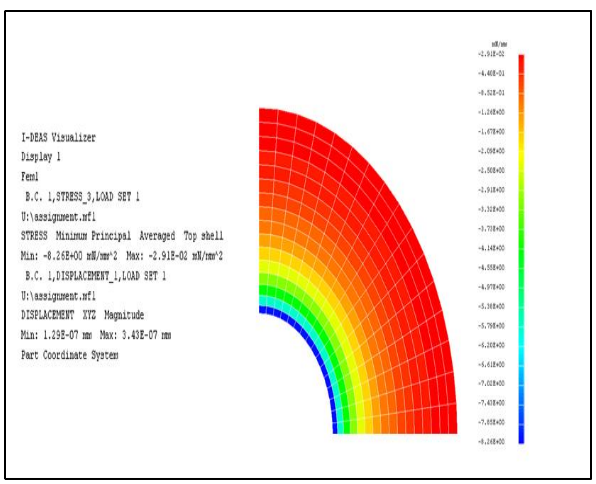
Implement the same steps which implemented in the sample (5*8) to get the result for the sample (16*16) as below.



Figure(7) shows the fully fixed restraint and the force after created



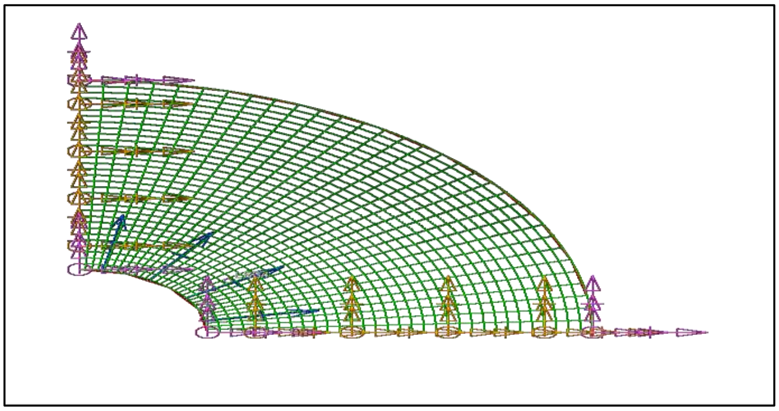
Figure(8) shows the Maximum Principal stress of (16*16)



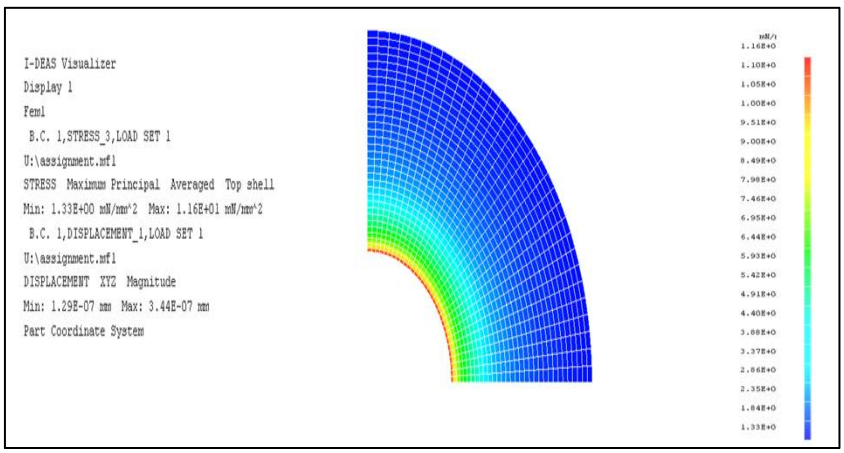
Figure(9) shows the Minimum Principal stress of (16*16)

3.Sample (32*32)

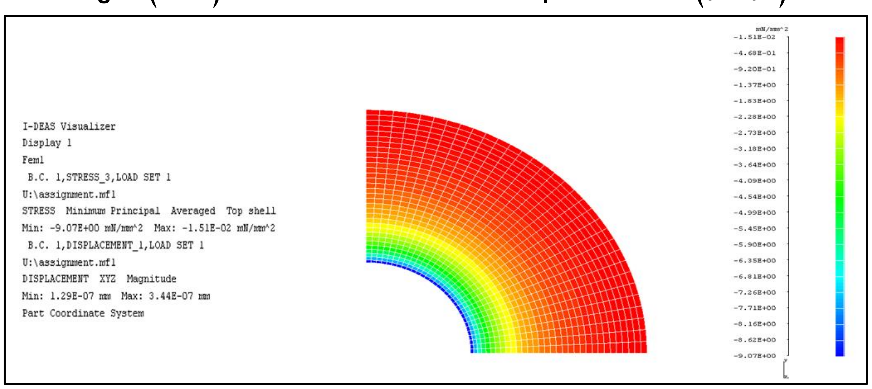
Implement the same steps which implemented in the s ample (5*8) to get the result for the sample (32*32) as below



Figure(10) shows the fully fixed restraint and the force after created



Figure(11) shows the Maximum Principal stress of (32*32).



Figure(12) shows the Minimum Principal stress of (32*32)

4.Sample (64*64)

Implement the same steps which implemented in the sample (5*8) to get the result for the sample (64*64) as shown.

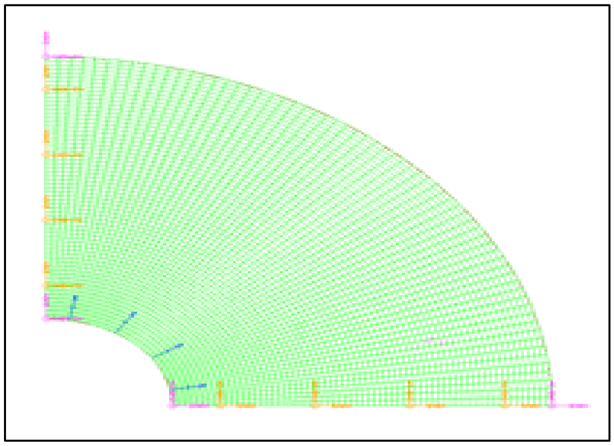
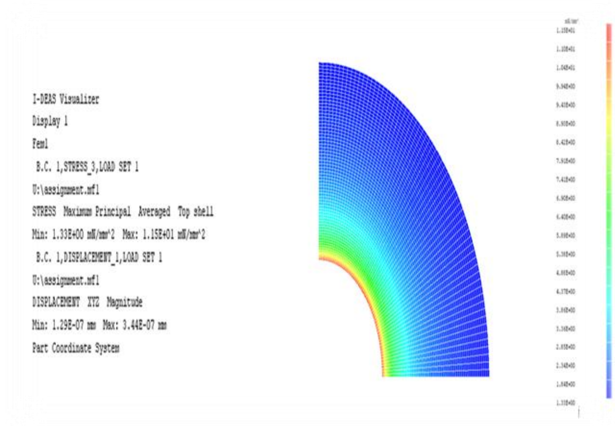
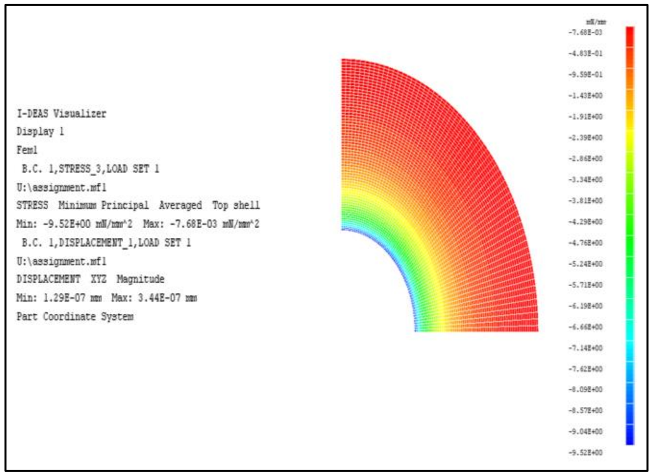


Figure (13) shows the fully fixed restraint and the force after created



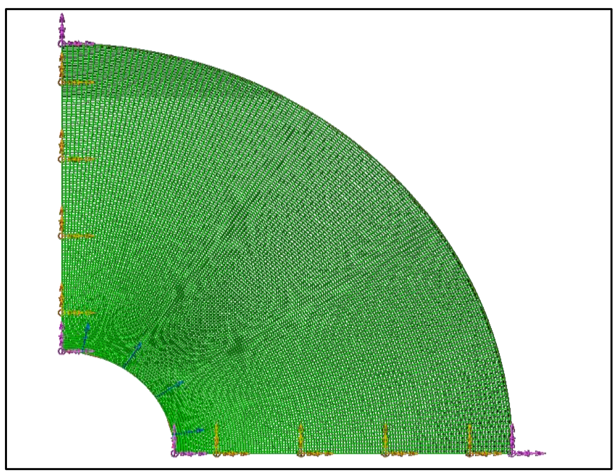
Figure(14) shows the Maximum Principal stress of (64*64)



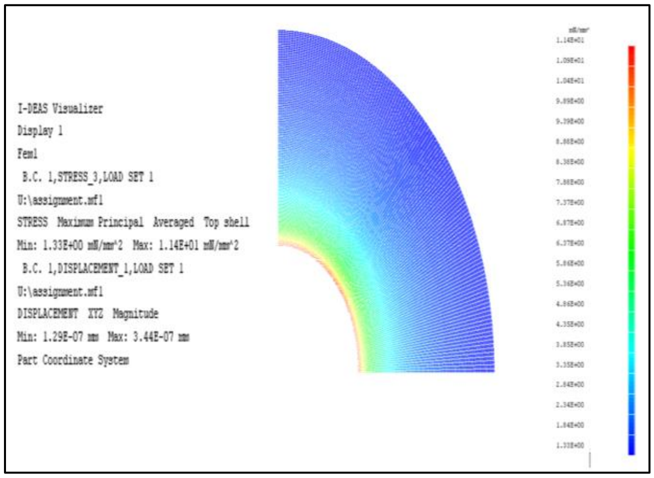
Figure(15) shows the Minimum Principal stress of (64*64)

5.Sample **(128*128)**

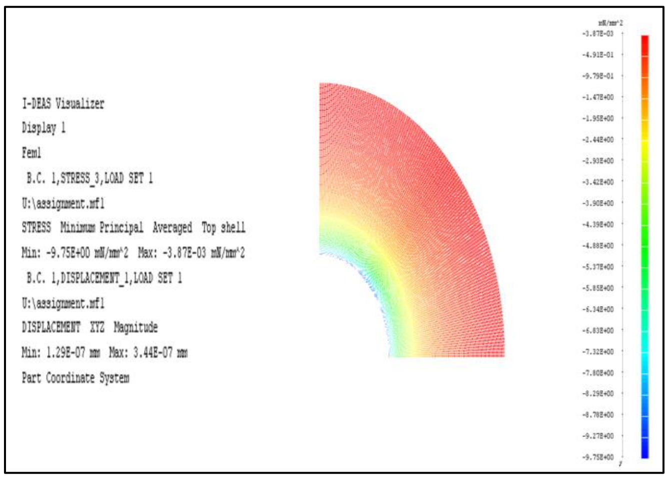
Implement the same steps which implemented in the sample (5*8) to get the result for the sample (128*128) as below



Figure(16) shows the fully fixed restraint and the force after created



Figure(17) shows the Maximum Principal stress of (128*128)

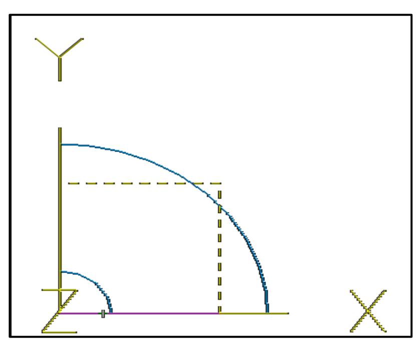


Figure(18) shows the Minimum Principal stress of (128*128)

Theoretical values :

In this case the handle calculations will shows the theoretical values for the hoop and radial stresses (σ h and σ r) respectively

Equations



$$\sigma r = A - \frac{B}{r^2} \qquad \text{Eq (1)}$$

$$-10 = A - \frac{1}{5^2}$$

$$\sigma h = A + \frac{B}{r^2} \qquad \text{Eq (2)}$$

Internal Radius (b) = 5mm , External Radius (a) = 20mm , Internal Radius (r) = 5mm

σ h = 0 N/m^2 , σ r = −10 N/mm^2

$-10 = A - \frac{1}{5^2}$ B=A−0.04B = 0.0025 **Eq (3)**

0 = A + —— B = A + 0.0025 B **Eq (4)**

Re−arrange (3) ⟶ A = 0.6675

Re−arrange (4) ⟶ B = 267

Inside radial

$\sigma h = A + \frac{B}{r^2} = 0.6675 + \frac{267}{5^2} = 11.3475$ N/mm^2

$\sigma r = A - \frac{B}{r^2} = -10$ N/mm^2

Outer radial

$\sigma h = A + \frac{B}{r^2} = 0.6675 + \frac{267}{5^2} = 1.335$ N/mm^2

$\sigma r = A - \frac{B}{r^2} = 0$ N/mm^2.

Through the comparison between the results from I−DEAS program and handle calculations that will see it from the table below the close result is the sample of (128 * 128).

Table (1) shows the comparison between hand calculations and I–DEAS Program

No	Sample Size	Hoop stress		Radial stress	
		Maximum Principal		Minimum Principal	
		Outer radial mN/mm^2	Inner radial mN/mm^2	Outer radial mN/mm^2	**Inner radial mN/mm^2**
1	**(5*8)**	1.25	11.3	−5.68	**$-9.79*10^{-2}$**
2	**(16*16)**	1.32	11.7	−8.26	**$-2.91*10^{-2}$**
3	**(32*32)**	1.33	11.6	−9.07	**$-1.51*10^{-2}$**
4	**(64*64)**	1.33	11.5	−9.52	**$-7.68*10^{-3}$**
5	**(128*128**	1.33	11.4	−9.75	**$-3.87*10^{-3}$**
6	**Hand Calculations**	**1.335**	**11. 3475**	**−10**	**0**

Clearly see from the table the hoop stress (σ h) for all samples are different .Also , by comparing between the results from I–DEAS we will note that the nearest result for the Hoop stress from hand calculations is the sample 128 * 128 because the hoop stress for 128 * 128
σ h = 11.4 N/mm2 is nearly equivalent for the hoop stress by hand calculations σ h = 11.3475 N/mm2 .

DISCUSSION:

1–The line graph below shows the relation between the outer radial(Maximum Principal) for the hoop stress from I–DEAS program and the outer radial for the hoop stress by hand calculations with different sizes meshes which are (5*80,(16*16),(32*32),(64*64) and (128*128).

Clearly see from the line graph the outer radial (Maximum Principal) for the hoop stress from I–DEAS increase from size mesh (5*8) to size mesh (128*128).However, the outer radial for the hoop stress by hand calculations is remained stable. Also, by comparing between all samples the nearest value for hand calculations is size mesh (128*128).

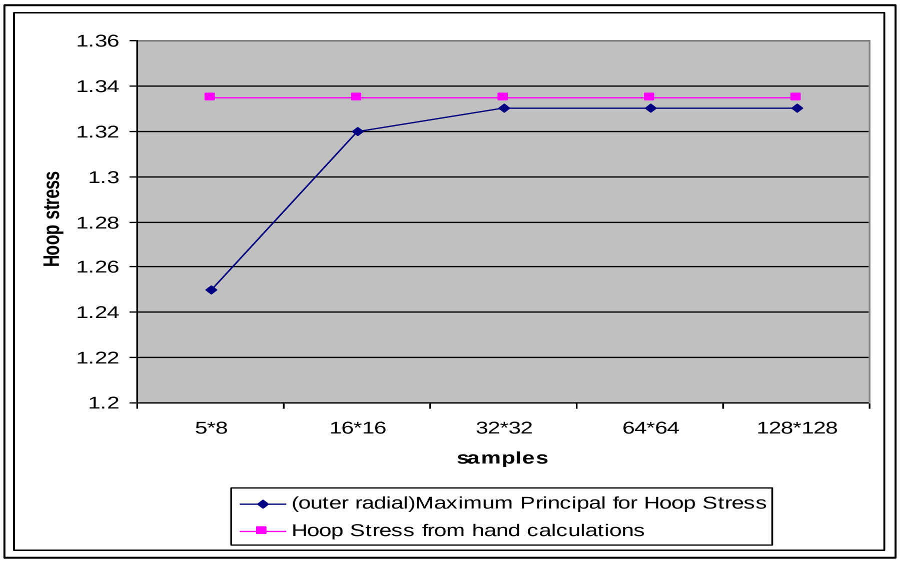
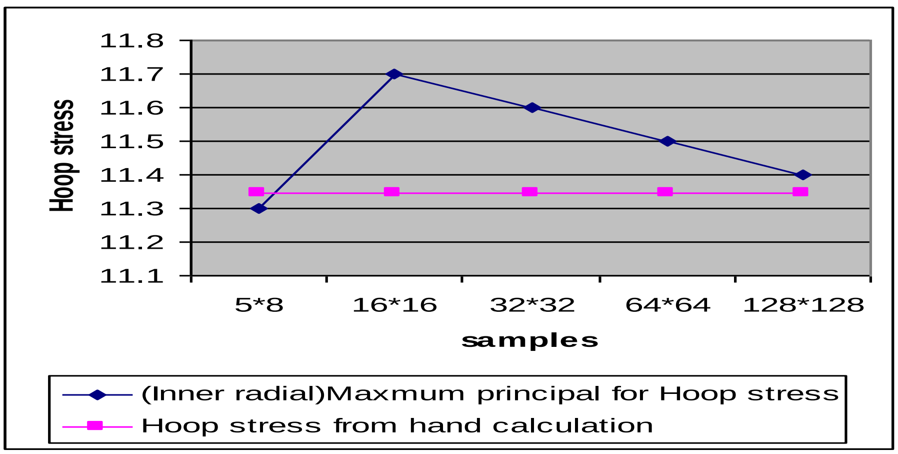


Figure (19)shows the relation between the outer radial for hoop stress from I-DEAS and the outer radial for hoop stress by hand calculations with different sizes meshes.

2-The line graph below shows the relation between the Inner radial(Maximum Principal) for the hoop stress from I-DEAS program and the Inner radial for the hoop stress by hand calculations with different sizes meshes which are (5*80,(16*16),(32*32),(64*64) and (128*128).

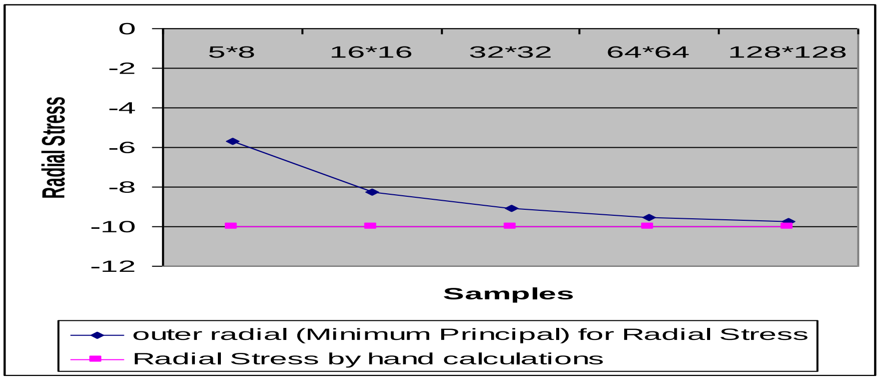
clearly see from the line graph the Inner radial (Maximum Principal) for the hoop stress from I-DEAS increase sharply from size mesh (5*8) to size mesh (16*16) then decrease gradually from size mesh (16*16) to size mesh (128*128) .However, the Inner radial for the hoop stress by hand calculations is remained stable. Also, by comparing between all samples the nearest value for hand calculations is size mesh (128*128).



Figure(20) shows the relation between the Inner radial for hoop stress from I-DEAS and the Inner radial for hoop stress by hand calculations with different sizes meshes.

3-The line graph below shows the relation between the outer radial(Minimum Principal) for the radial stress from I-DEAS program and the outer radial for the radial stress by hand calculations with different sizes meshes which are (5*85),(16*16),(32*32),(64*64) and (128*128).

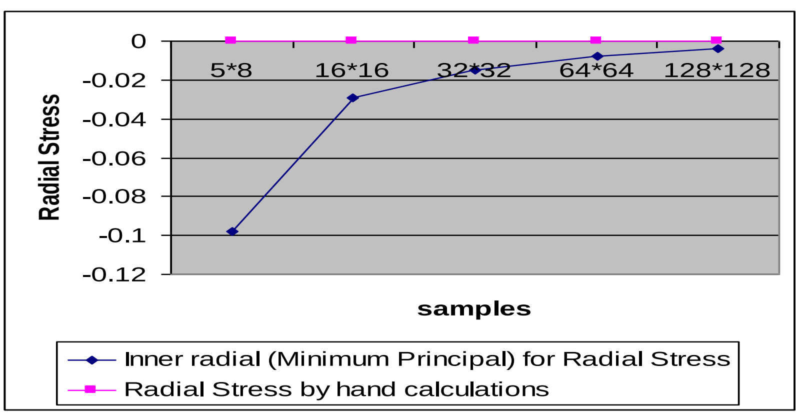
Clearly see from the line graph the outer radial (Minimum Principal) for the radial stress from I-DEAS decrease from size mesh (5*8) to size mesh (128*128).However, the outer radial for the radial stress by hand calculations is remained stable. Also, by comparing between all samples the nearest value for hand calculations is size mesh (128*128).



Figure(21) shows the relation between the outer radial for radial stress from I-DEAS and the outer radial for radial stress by hand calculations with different sizes meshes.

4-The line graph below shows the relation between the Inner radial(Minimum Principal) for the radial stress from I-DEAS program and the Inner radial for the radial stress by hand calculations with different sizes meshes which are (5*80,(16*16),(32*32),(64*64) and (128*128).

Clearly see from the line graph the Inner radial (Minimum Principal) for the radial stress from I-DEAS increase from size mesh (5*8) to size mesh (128*128).However, the Inner radial for the radial stress by hand calculations is remained stable. Also, by comparing between all samples the nearest value for hand calculations is size mesh (128*128).



Figure(22) shows the relation between the Inner radial for radial stress from I-DEAS and the Inner radial for radial stress by hand calculations with different sizes meshes

Conclusion:

During this research, it has been studied the finite element analysis for halve wall cylinder with various mesh size, it was using two methods that are explained above, noticed that the comparison between the methods for result of hoop and radial stress by I-DEAS and hand calculation by using the Equation. As explained above that The results were gotten compression for all mesh size, from here noticed that the hoop stress from I-DEAS increase from size mesh (5*8) to size mesh (128*128). So in the hand calculation is remained stable. Also through comparing between all samples that the nearest value for hand calculations is size mesh (128*128).

References

1. The tutorial from the help library of the I-DEAS Software.
2. tress analysis of compound cylinders subjected to thermomechanical loads". J. Mechanical Science and Technology, Vol.31, Issue 4, PP. 1805-1811.
3. Lohar Hariram, 2013, "Steess analysis and brust pressure determination of two layered compound cylinder". IJEST Vol 5.
4. Vasko Tajana , Franjo., 2014 "Stress and deformation analysis of compound cylinders in elastic plastic range" , PhD Thesis, University of Osijek
5. (Warren C. Young, Richard G Budynas , Roark's Formulas for Stress and Strain,2002, McGrawHill 3. Narasaiha, L. (2008). Finite Element Analysis. CRC..
6. S. Xu & M. Yu, "Shakedown Analysis of thick walled cylinders subjected to internal pressure with inified strength criterion", International Journal of Pressure vessels & piping, Vol. 82, pp706- 712,2005. 8. Y. Z. Chen & X. Y. Lin, "An alternative numerical solution of thick walled cylinders and spheres made of functionally graded materials", Computational Materials Science, vol. 48, pp640-647, 2010 .
7. Choudhury Susanta, Roy H., 2010 "Stress analysis of a thick walled cylinder". B Tech thesis, National Institute of Technology India
8. K. Vedeld, H. A. Sollund and J. Hellesland, 2015,"Closed analytical expressions for stress distributions in two-layer cylinders and their application to offshore lined and clad pipes", Journal of Offshore Mechanics and Arctic Engineering, Vol. 137, Isuue 2 PP. 021702-021702-9
9. Y. Z. Chen & X. Y. Lin, "An alternative numerical solution of thick walled cylinders and spheres made of functionally graded materials", Computational Materials Science, vol. 48, pp640-647, 2010