Chapter 4 Theoretical-numerical analysis of the stress concentration factor in a stepped bar in tension

Capítulo 4 Análisis teórico-numérico del factor de concentración de esfuerzos en una barra escalonada sometida a tracción

SOTO-MENDOZA, Gilberto*, MARTÍNEZ-GARCÍA, José and COUTIÑO-MORENO, Elvis

Tecnológico Nacional de México. Tecnológico de Estudios Superiores de Jocotitlán. Departamento de Ingeniería Mecatrónica. Carretera Toluca-Atlacomulco KM 44.8, Ejido de San Juan y San Agustín, Jocotitlán, 50700 México.

ID 1st Author: *Gilberto, Soto-Mendoza* / **ORC ID**: 0000-0001-7357-9445, **CVU CONAHCYT ID**: 635154

ID 1st Co-author: *José, Martínez-García* / **ORC ID**: 0000-0002-7797-1062, **CVU CONAHCYT ID**: 612069

ID 2nd Co-author: *Elvis, Coutiño-Moreno* / **ORC ID**: 0000-0003-2455-2574, **CVU CONAHCYT ID**: 550285

DOI: 10.35429/H.2023.5.47.60

G. Soto, J. Martínez and E. Coutiño

*gilberto.soto@tesjo.edu.mx

R. López (AA.) Engineering and Architecture in the Northern part of the State of Mexico. Handbooks-TI-©ECORFAN-Mexico, Estado de México, 2023

Abstract

The stress concentration factor of different structural elements has been obtained analytical, experimental, and numerically. For the analytical part, there are several formulas established according to the element and the load condition to which the component is subjected. In the case of the numerical simulation results, they are approximations that depend on the refinement of the mesh. In this work, a comparative analysis was carried out between the analytical results and those obtained numerically from the stress concentration factor of a flat stepped bar subjected to tension, considering convergence criteria of 5%, 2% and 1%. An automatic mesh refinement tool was used, and several studies were run using design points. It was found that the average percentage variation between the analytical and the numerical approach according to the convergence criterion was 2.5%, 0.9% and 0.6%, respectively. Also, some points were found where the variation was notable.

Stress concentration, FEM, Simulation, ANSYS

Resumen

El factor de concentración de tensiones de diferentes elementos estructurales se ha obtenido analítica, experimental y numéricamente. Para la parte analítica, existen diversas fórmulas establecidas en función del elemento y de la condición de carga a la que está sometido el componente. En el caso de los resultados de simulación numérica, son aproximaciones que dependen del refinamiento de la malla. En este trabajo se realizó un análisis comparativo entre los resultados analíticos y los obtenidos numéricamente del factor de concentración de tensiones de una barra plana escalonada sometida a tracción, considerando criterios de convergencia del 5%, 2% y 1%. Se utilizó una herramienta de refinamiento automático de la malla y se realizaron varios estudios utilizando puntos de diseño. Se comprobó que la variación porcentual media entre el enfoque analítico y el numérico según el criterio de convergencia era del 2,5%, 0,9% y 0,6%, respectivamente. Además, se encontraron algunos puntos en los que la variación era notable.

Concentración de tensiones, MEF, Simulación, ANSYS

1. Introduction

In the field of engineering, when designing components such as shafts, supports, gears, among others, factors such as the concentration of efforts must be considered. Three different approaches can be used to determine these factors. The first is theoretical analysis. This is limited when dealing with complex geometries for which there are no established formulas or are very complex that it is not practical to use. The second is the experimental analysis, which is expensive to implement and the time to perform it is considerable. The third approach is simulation. The latter solves the governing equations of the physical phenomenon to be treated and has been implemented for the solution of structural, thermal, fluid mechanics problems, among others.

A reference author who has developed a series of formulas to obtain the concentration of stresses of several structural elements subject to different load conditions is Roark (Budynas & Sadegh, 2020). In his publication he presents a summary of formulas, facts and principles related to mechanics of materials. It contains a series of tables to obtain the stress concentration factor.

On the other hand, in the experimental approach, stress concentration studies using photoelasticity are reported. Among these studies Weibel (Studies in Photoelastic Stress Determination | J. Fluids Eng. | ASME Digital Collection, 1934) reports three laboratory studies to determine the concentration of efforts. Wilson and White (Wilson & White, 1973) use the method to analyze concentration of stresses generated by fillets and grooves in plates subjected to tension and bending. Other studies report the combination of the approaches mentioned for obtaining the stress concentration factor, for example, Ronald and Bastida (Roldan & Bastidas, 2002) report a study of the stress concentrator factor produced by a hole in a flat plate.

Regarding numerical simulation, this provides approximate solutions and is based on the Finite Element Method. In this methodology, geometry is divided into parts as if they were the pieces of a LEGO® (Chen & Liu, 2018). To represent the curves, smaller elements are required to define them better. The use of this approach has been potentiated due to the improvement in computer equipment and that compared to experimental studies represents a lower cost. Comparative studies have been carried out between analytical and numerical results. For example, Chmelko *et al* (Chmelko *et al.*, 2021) report a theoretical-numerical study focused on the analysis of the concentration of effort that are presented in notches considering different mesh sizes.

This paper aims to analyze the variation of the stress concentration factor in a tension step bar obtained with theoretical calculation and numerical analysis using automatic mesh refinement tools, to find the percentage variation under different convergence criteria.

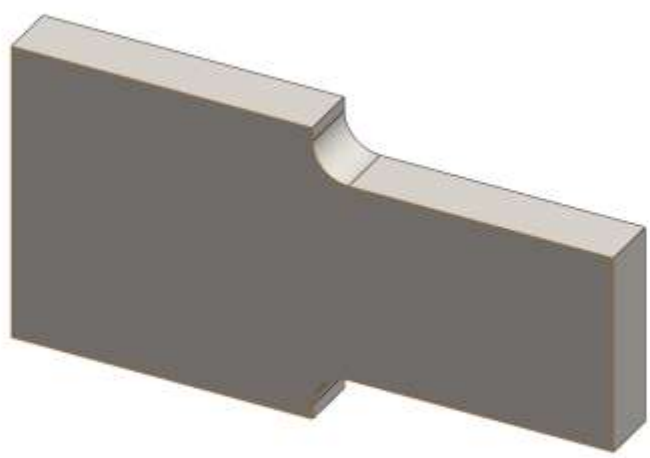
2. Description of the problem

The concentrations of stresses that occur in structural components of machines, land vehicles, aircraft, etc. cause that in the area where they are located, the stress resisted by the material when applying load is quickly reached. For this reason, it is important to have your values well defined. This can be carried out theoretical, experimentally, or numerically. The results obtained by the different approaches should be the same or very similar.

Currently, numerical simulation has become a very powerful tool in engineering. However, certain principles must be followed to validate that the result obtained is correct. In the case of structural static analysis, a mesh sensitivity analysis must be carried out, that is, a refinement of the mesh and that the desired result is the same regardless of the mesh size used.

It can be said that the theoretical solution is the exact or ideal solution and the numerical result is an approximate solution that will depend on the discretization of the component. There is a convergence tool that automatically performs the mesh refinement process considering a convergence criterion. In this paper, to find the percentage variation between theoretical analysis and numerical simulation considering different convergence criteria of a stepped flat bar (see Figure 1).

Figure 1 Stepped flat bar



3. Methodology

Once the type of element to be studied (stepped flat bar) was defined, to obtain the percentage of variation between the results of the theoretical and numerical approach, the following stages were followed:

1. Creation of geometry respecting the principle of Saint-Venant.
2. Theoretical calculation of the effort concentrator according to Roark.
3. Numerical simulation using a program CAE (Computer Aided Engineering).
4. Comparative analysis of results obtained theoretically with those obtained numerically.

The next section begins with the theoretical basis of the case study.

4. Theoretical basis

4.1. Stress Concentrations

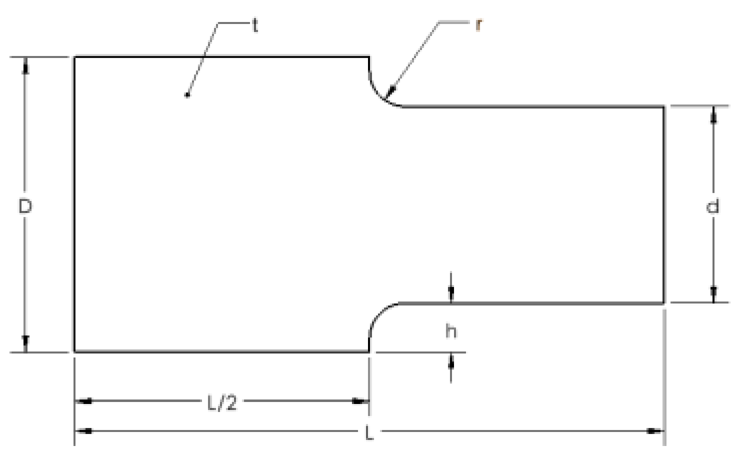
Stress concentrations occur in regions of the components where fillets, grooves, holes, or some other change in geometry are found. To determine the maximum stress that is presented, the Equation 1 is used. For more information it is recommended to consult the references (Ferdinand *et al.*, 2020; Goodno & Gere, 2018; R. C. Hibbeler, 2018).

$$\sigma_{máx} = K\sigma_{nom} \quad (1)$$

Where:
$\sigma_{máx}$ = Maximum stress
K = Stress Concentration Factor
σ_{nom} = Nominal stress

For a stepped flat bar (see Figure 2), it was found that the stress concentration factor is determined with Equation 2, according to Roark (Budynas & Sadegh, 2020):

Figure 2 Stepped flat bar with dimensions



$$K = C_1 + C_2\left(\frac{2h}{D}\right) + C_3\left(\frac{2h}{D}\right)^2 + C_4\left(\frac{2h}{D}\right)^3 \quad (2)$$

To get the constants you must:

If $0.1 \leq h/r \leq 2.0$

$$C_1 = 1.007 + 1.000\sqrt{\frac{h}{r}} - 0.0031\frac{h}{r} \quad (3)$$

$$C_2 = -0.114 - 0.585\sqrt{\frac{h}{r}} + 0.314\frac{h}{r} \quad (4)$$

$$C_3 = 0.241 - 0.992\sqrt{\frac{h}{r}} - 0.271\frac{h}{r} \quad (5)$$

$$C_4 = -0.134 + 0.577\sqrt{\frac{h}{r}} - 0.012\frac{h}{r} \quad (6)$$

If $2.0 \leq h/r \leq 20.0$

$$C_1 = 1.007 + 1.000\sqrt{\frac{h}{r}} - 0.0031\frac{h}{r} \quad (7)$$

$$C_2 = -0.114 - 0.585\sqrt{\frac{h}{r}} + 0.314\frac{h}{r} \quad (8)$$

$$C_3 = 0.241 - 0.992\sqrt{\frac{h}{r}} - 0.271\frac{h}{r} \quad (9)$$

$$C_4 = -0.134 + 0.577\sqrt{\frac{h}{r}} - 0.012\frac{h}{r} \quad (10)$$

4.2. Numerical analysis

In general, the steps to perform a static structural numerical analysis are:

- Definition of the mechanical properties of the material.
- Mesh generation.
- Application of border conditions.
- Study solution.
- Obtaining the desired results such as deformation, efforts, etc. (Postprocessing).

Equation 11 corresponds to the governing equation for a linear structural static study.

$$[\boldsymbol{K}]\{\boldsymbol{D}\} = \{\boldsymbol{F}\} \quad (11)$$

Where:

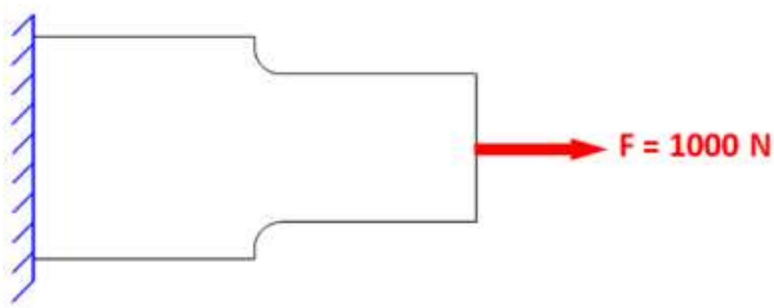
$\{D\} =$ = Displacement vector
$\{F\}$ = Force vector
[K] = The stiffness matrix

5. Theoretical-numerical analysis

5.1. Theoretical analysis

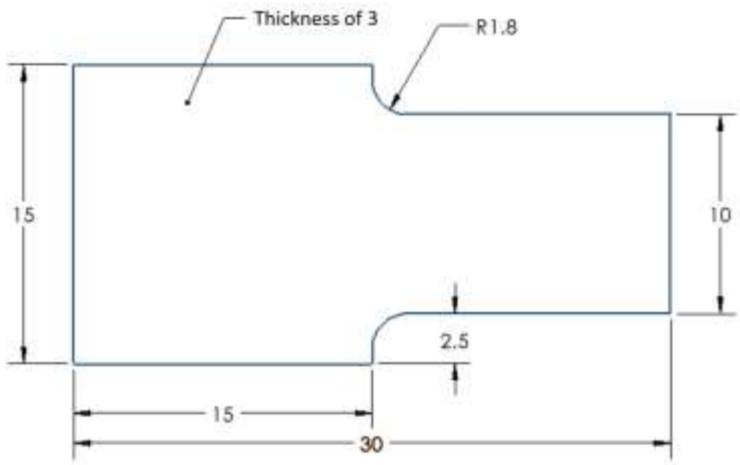
This section shows the theoretical calculation of the maximum stress produced by the stress concentration of the flat step bar and the different stress concentration factors if the dimensions of the part are varied. Figure 3 shows a flat stepped bar recessed at its left end and a force of 1000 N is applied on its right side.

Figure 3 Support and load applied to the flat stepped bar



The dimensions of the part are illustrated in Figure 4. The dimensions are given in millimeters. For the construction of the geometry was considered the principle of Saint-Venant which says that the equation σ = F/A defines axial stresses on a cross section of the bar only when the section is at least one distance B away from any concentrated load or discontinuity in its shape, where B is the largest lateral dimension of the bar (Goodno & Gere, 2018).

Figure 4 Dimensions of the flat step bar



To obtain the stress concentrator the equation 2 is used. It contains four constants: C_1, C_2, C_3 y C_4. Which are calculated with equations 3 to 10. To define the equations to be used, the h/r ratio is calculated. From Figure 4 we have that h = 2.5 mm and r = 1.8 mm, therefore, h/r = 2.5 mm/1.8 mm = 1.389. This indicates that the equations to be used to obtain the constants are from 3 to 6. Substituting the numerical values into the equation yields:

If $0.1 \leq 1.667 \leq 2.0$

$$C_1 = 1.007 + 1.000\sqrt{\frac{2.5}{1.8}} - 0.0031\frac{2.5}{1.8} = 2.143$$

$$C_2 = -0.114 - 0.585\sqrt{\frac{2.5}{1.8}} + 0.314\frac{2.5}{1.8} = -0.3673$$

$$C_3 = 0.241 - 0.992\sqrt{\frac{2.5}{1.8}} - 0.271\frac{2.5}{1.8} = -1.3045$$

$$C_4 = -0.134 + 0.577\sqrt{\frac{2.5}{1.8}} - 0.012\frac{2.5}{1.8} = 0.5293$$

Substituting the values into equation 2 gives:

$$K = 2.143 + (-0.3673)\left[\frac{(2)(2.5)}{15}\right] + (-1.3045)\left[\frac{(2)(2.5)}{15}\right]^2 + (0.5293)\left[\frac{(2)(2.5)}{15}\right]^3 = 1.895$$

The maximum stress is now calculated. The force applied to the part is 1000 N. The nominal effort corresponds to the section on the right side (see Figure 4). The cross section is rectangular. Using Equation 2 we have:

$$\sigma_{nom} = \frac{1000\ N}{(10\ mm)(3\ mm)} = 33.33\ MPa$$

$$\sigma_{máx} = (1.895)[33.33\ MPa] = 63.2\ MPa$$

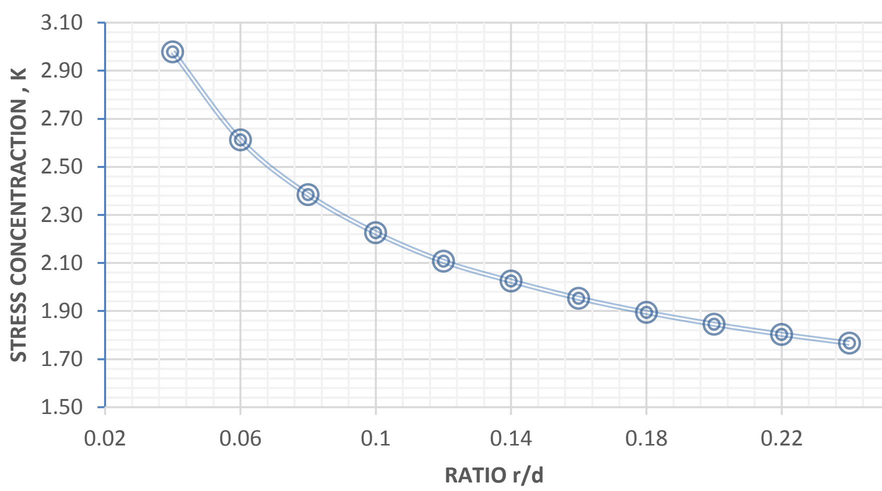
The above procedure is applied to obtain the stress concentrator when the radius of Figure 4 is modified. The other dimensions remain constant. A spreadsheet was created for efficiency. Table 1 shows the results of the stress concentrator and the maximum stress of the flat step bar for different radius dimensions.

Table 2 Maximum stress and stress concentrator for different radius

No.	Radio, r (mm)	Ratio r/d	Stress concentrator, K	Maximum stress, σ_{max} (MPa)
1	0.4	0.04	2.98	99.3
2	0.6	0.06	2.61	87.1
3	0.8	0.08	2.38	79.5
4	1	0.1	2.23	74.2
5	1.2	0.12	2.11	70.2
6	1.4	0.14	2.02	67.5
7	1.6	0.16	1.953	65.1
8	1.8	0.18	1.895	63.2
9	2	0.2	1.845	61.5
10	2.2	0.22	1.803	60.1
11	2.4	0.24	1.767	58.9

Derived from Table 1, Figure 5 is obtained. This shows on the vertical axis the effort concentrator and on the horizontal axis the ratio r/d, for a ratio D/d = 15 mm/10 mm = 1.5.

Figure 5 Stress concentration K vs ratio r/d for D/d = 1.5.

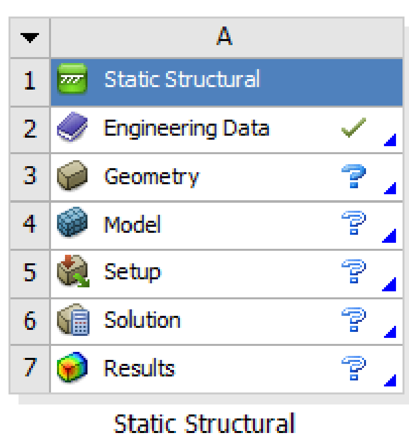


5.2. Numerical analysis

The numerical study was conducted in the ANSYS Student program, version 2023/R1. This is a free academic license. Which is limited to the number of elements or nodes that can solve, in particular, for structural analyses the limit is 128,000 nodes or elements, whichever is reached first. To download this package, consult the installation procedure, system requirements, license duration, among others, see the reference (ANSYS, 2023).

Below are each of the stages required for a Static-Structural study in the package. ANSYS (see Figure 6). The first cell A1, indicates the type of study “Static Structural”.

Figura 6 Structural Static Module in ANSYS Workbench



5.2.1 Material properties

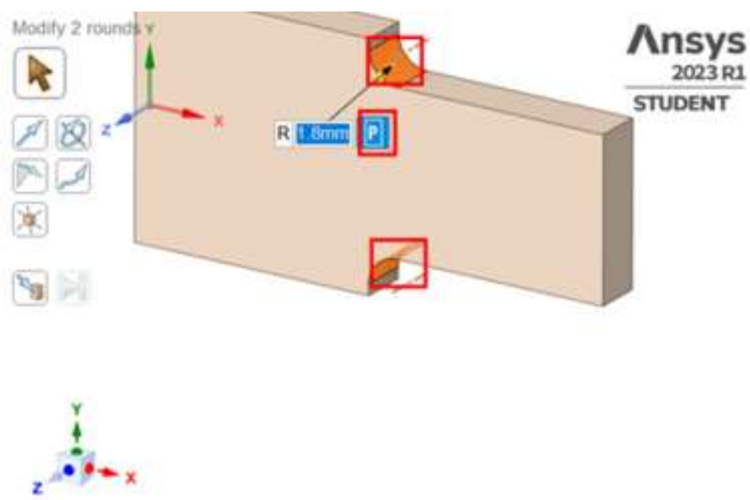
The second cell A2 of the workflow for the Static-Structural simulation corresponds to the materials "Engineering Data". In this section you can select from the library the material or materials to be used. If they do not exist, they can be created according to the properties of the material to be used. For the case study, a structural steel is used that has the following properties:

- Young's modulus = 200 GPa
- Poisson's ratio = 0.3
- Yield Strength = 250 MPa

5.2.2 Geometry

The geometry (cell A3) can be imported or created in the Computer Aided Design (CAD) programs of the same package. The available options integrated into the program are: SpaceClaim, DesingModeler and Discovery. In the present work, the SpaceClaim program was used to create the piece. In addition to this, the radius was parameterized, which allows several case studies to be carried out at a later stage. Figure 6 shows the part created. A red rectangle shows the radii to be parameterized and activating the box with a letter P activates the parameterization. When parameterizing either the geometry or some result, a box is created with the name of "Parameter Set".

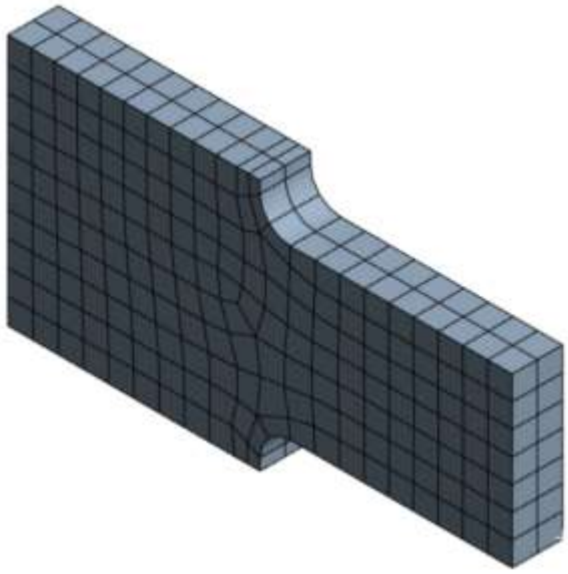
Figure 7 Geometry created in SpaceClaim



5.2.3 Mesh generation

The generation of the mesh was carried out in the program ANSYS Meshing. This is integrated into the cell module A4 Model. Related, to have 2 elements in the thickness of the piece in the global parameters of the mesh handles a mesh size of 1.5 mm. With an adaptive mesh and all other parameters are left by default. Obtaining a mesh with 2230 nodes, 353 elements and a minimum Element Quality of 0.66. Figure 8 shows the mesh created. This is an initial mesh for the study, since subsequently, a mesh sensitivity analysis is performed.

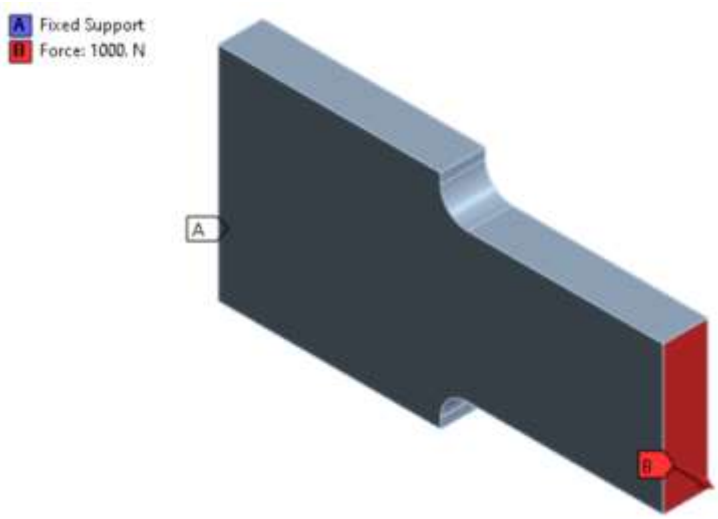
Figure 8 Mesh



5.2.4 Boundary conditions

Boundary conditions are a fixed support on the far-left side and a force of 1000 N on the right side. Figure 9 shows the boundary conditions of the part envelope. The blue color with the label "A" corresponds to the fixed support. The red color corresponds to the face where the force is applied and labeled with the letter “B”.

Figure 9 Boundary conditions



5.2.5 Postprocesamiento

Once the material has been selected and the boundary conditions established, the study is sent for solution. As results, displacement and normal stress in the X direction were considered. Once the simulation was executed, the results illustrated in Figure 10 and Figure 11 were obtained. Figure 10 shows the displacement in the X direction, its maximum magnitude is represented in red with a value of 0.00425 mm. Figure 11 illustrates that the stress is concentrated in the radius (red zone) with a magnitude of 56.69 MPa.

Figure 10 Displacement in the X direction.

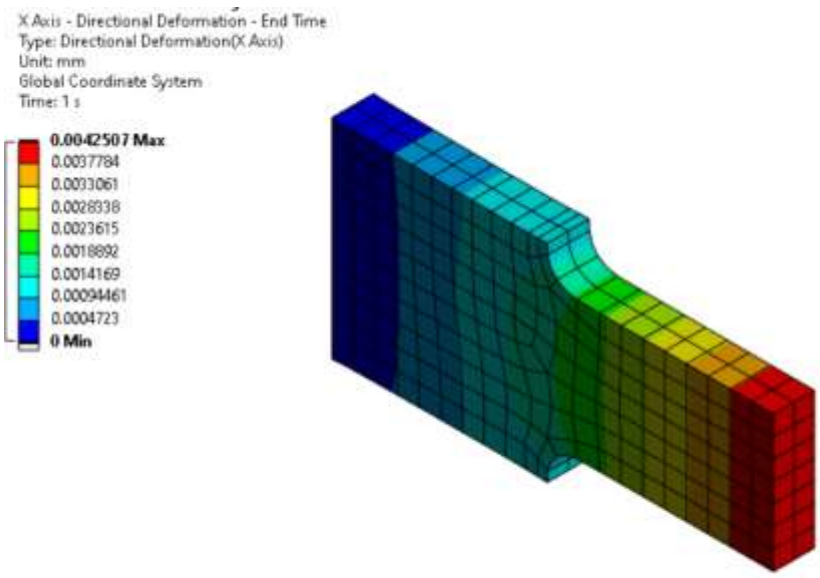
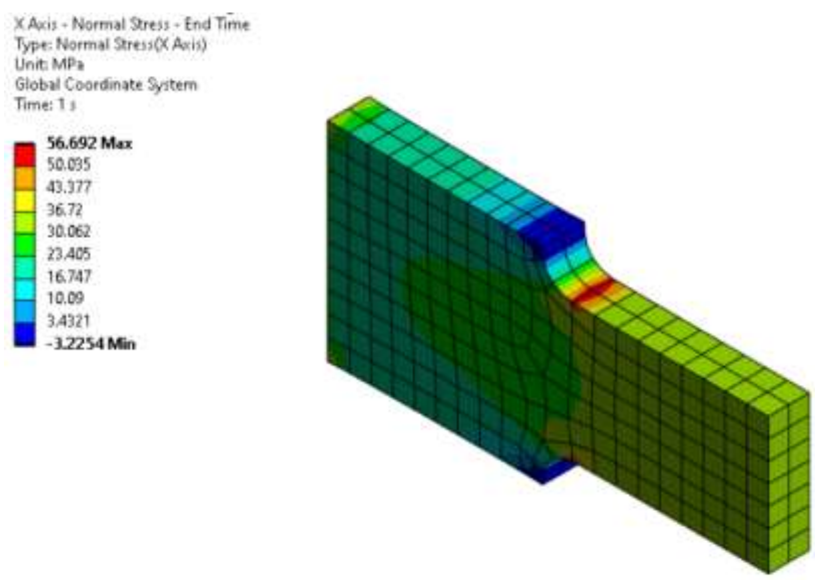


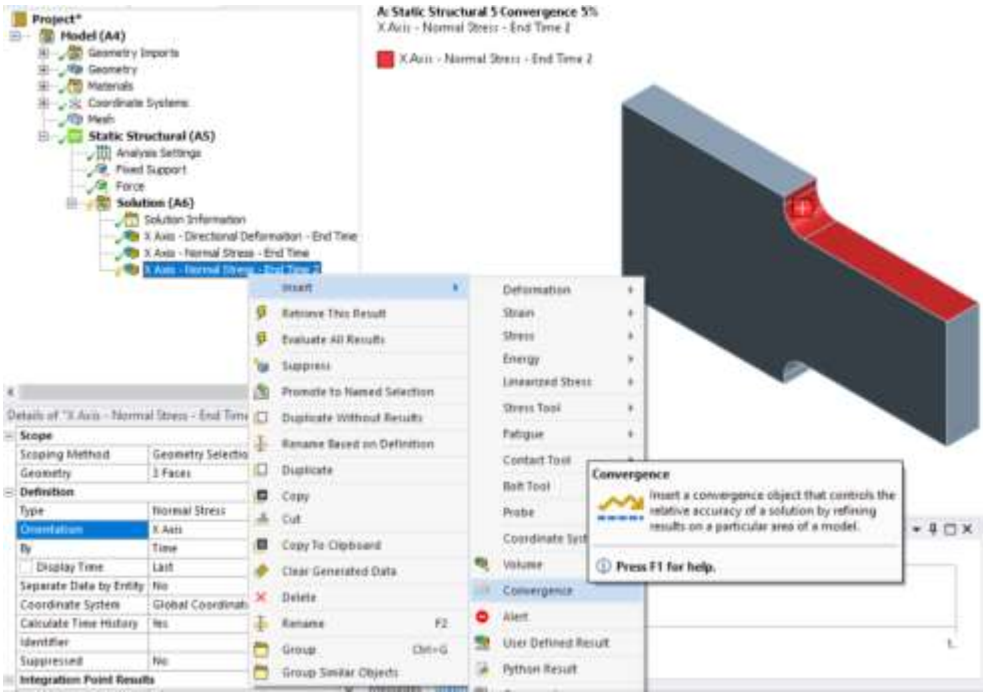
Figure 11 Normal stress in the X direction



5.2.6 Mesh sensitivity analysis

Since the results, in particular the efforts, vary with the size of the elements of the mesh, it is necessary to consider what is the acceptable percentage that indicates that convergence is achieved. Three are considered in this paper. The first when convergence is reached at 5%, the second when convergence is reached at 2% and the third when convergence is reached at 1%. One way to refine the mesh is to do it manually or use a tool "Convergence" (see Figure 12). In addition, the area of interest is where the radius is located, for this reason the face of the radius and the adjacent faces (red area) are selected.

Figure 12 Tool “Convergence”



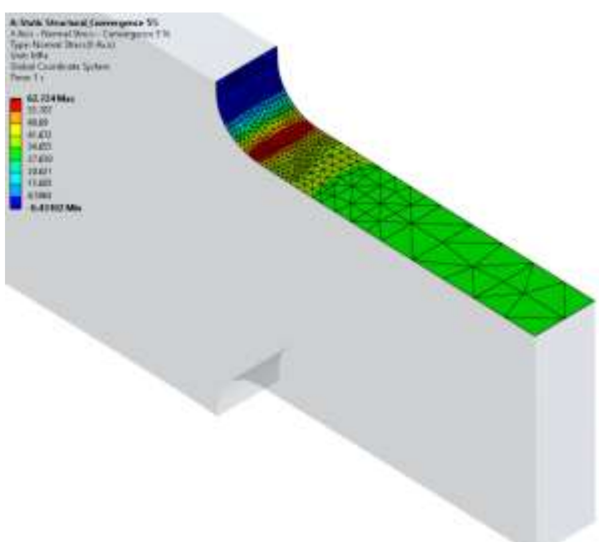
Considering a convergence criterion of 5% and executing the solution, the results shown in Table 2 are obtained. The first result obtained was 56.69 MPa with 2230 nodes and 358 elements. The convergence was reached in the third run obtaining an Stress of 62.72 MPa with 29787 nodes and 20012 elements. Between runs 2 and 3 the change in the result was -0.47%, so it meets the convergence criterion that indicates that the variation must be less than 5%.

Table 3 Mesh sensitivity analysis

Run	Normal Stress in the X direction (MPa)	Change (%)	Nodes	Elements
1	56.69		2230	358
2	63.02	8.838	13480	8374
3	62.72	-0.4702	29787	20012

Figure 13 shows the stress results in the area of interest. Also, the change in the size of the mesh elements can be observed.

Figure 13 Stress in the area of interest



5.2.7 Design Points

To obtain the stress for different radii considering a convergence of the results of 5%, 2% and 1%, at least 20 more runs have to be made. For this purpose, design points were used. Figure 14 shows in column B the different radii considered. Columns C, D and E show the Stress obtained considering a convergence of 5%, 2% and 1% respectively. It is important to note that in the case of cell E3 the result was not obtained because the refinement of the mesh reached the number of nodes that allows solving the academic license.

Figure 14 Design points

Table of Design Points

	B	C	D	E
1	P6 - Radio (r)	P7 - X Axis - Normal Stress - Convergence 5 %	P9 - X Axis - Normal Stress - Convergence 2 %	P10 - X Axis - Normal Stress - Convergence 1 %
2	mm	MPa	MPa	MPa
3	0.4	94.785	99.725	99.725
4	0.6	86.316	86.992	86.992
5	0.8	79.75	79.75	79.75
6	1	74.512	74.512	74.512
7	1.2	71.15	71.15	70.85
8	1.4	60.468	67.291	67.541
9	1.6	64.535	64.965	62.724
10	1.8	62.724	57.686	62.724
11	2	60.449	60.97	60.97
12	2.2	59.857	59.437	59.437
13	2.4	57.893	58.009	58.009
*				

Table 3 shows the numerically obtained stresses and the stress concentration factor. To obtain the stress concentration factor, the numerical stress is divided by the nominal stress that for the case study is 33.33 MPa.

Table 4 Stress concentration factor derived from numerical simulation

No.	Radio, r (mm)	Convergence 5%		Convergence 2%		Convergence 1%	
		Maximum Stress, σ_{max} (MPa)	Stress concentration, K	Maximum Stress, σ_{max} (MPa)	Stress concentration, K	Maximum Stress, σ_{max} (MPa)	Stress concentration, K
1	0.4	94.79	2.84	99.73	2.99	-	-
2	0.6	86.32	2.59	86.99	2.61	86.99	2.61
3	0.8	79.75	2.39	79.75	2.39	79.75	2.39
4	1	74.51	2.24	74.51	2.24	74.51	2.24
5	1.2	71.15	2.13	71.15	2.13	70.85	2.13
6	1.4	60.47	1.81	67.29	2.02	67.54	2.03
7	1.6	64.54	1.94	64.97	1.95	62.72	1.88
8	1.8	62.72	1.88	57.69	1.73	62.72	1.88
9	2	60.45	1.81	60.97	1.83	60.97	1.83
10	2.2	59.86	1.80	59.44	1.78	59.44	1.78
11	2.4	57.89	1.74	58.01	1.74	58.01	1.74

6. Analysis of results

In this section, a comparative analysis is carried out between the theoretical result of the stress concentration factor of a stepped flat bar and this same using numerical simulation with three different convergence ranges (5%, 2% and 1%). From Table 1 the results of the stress concentration factor obtained theoretically are extracted and from Table 3 the numerical values. Table 4 summarizes the values of concentration of stress and their percentage variation with respect to the theoretical value.

Table 5 Percentage variation of theoretical-numerical stress concentrations factors

No.	Radio, r (mm)	Theoretical stress concentration factor, K	Convergence 5%		Convergence 2%		Convergence 1%	
			Stress concentration factor, K	Variation (%)	Stress concentration factor, K	Variation (%)	Stress concentration factor, K	Variation (%)
1	0.4	2.98	2.84	4.5	2.99	-0.4	-	-
2	0.6	2.61	2.59	0.8	2.61	0.1	2.61	0.1
3	0.8	2.38	2.39	-0.3	2.39	-0.3	2.39	-0.3
4	1	2.23	2.24	-0.4	2.24	-0.4	2.24	-0.4
5	1.2	2.11	2.13	-1.3	2.13	-1.3	2.13	-0.9
6	1.4	2.02	1.81	10.4	2.02	0.3	2.03	-0.1
7	1.6	1.95	1.94	0.9	1.95	0.2	1.88	3.7
8	1.8	1.89	1.73	8.7	1.73	8.7	1.88	0.7
9	2	1.85	1.81	1.7	1.83	0.9	1.83	0.9
10	2.2	1.80	1.80	0.4	1.78	1.1	1.78	1.1
11	2.4	1.77	1.74	1.7	1.74	1.5	1.74	1.5

From Table 4 we must:

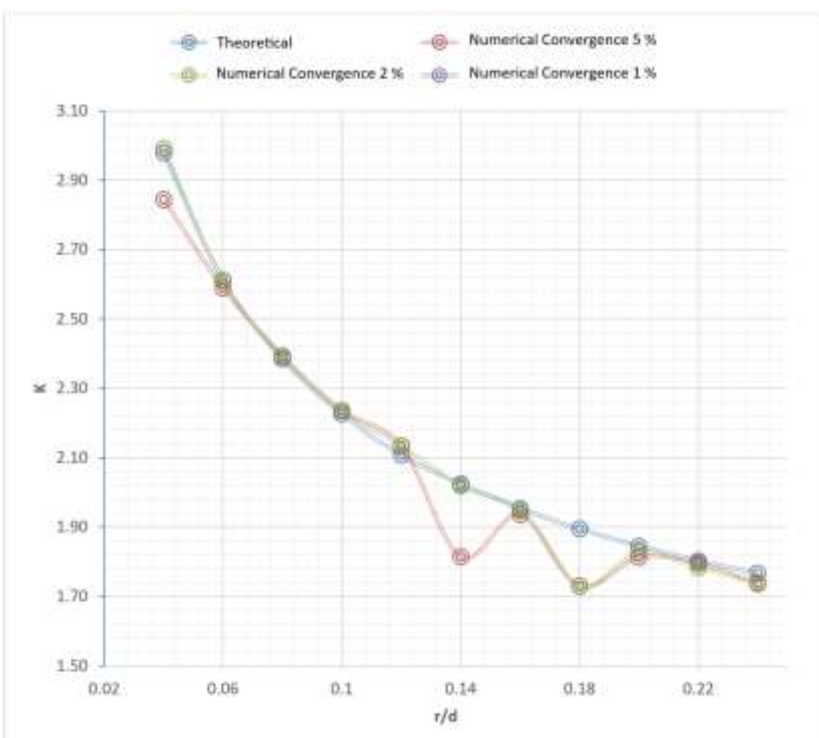
– The average of the percentage of fluctuation considering a convergence range of 5% is 2.5% and the maximum variation is 10.4%.

– The average of the percentage fluctuation considering a convergence range of 2% is 0.9% and the maximum variation is 8.7%.

– The average of the percentage of fluctuation considering a convergence range of 1% is 0.6% and the maximum variation is 3.7%.

As the magnitude of the convergence criterion is reduced, that is, the percentage of variation of the effort due to mesh refinement is lower, the numerical and theoretical results are closer. However, the number of nodes and elements increases, and this is reflected in a greater demand for computational resources. In the case of the academic license, it is limited to the permitted limit of nodes or elements mentioned in point 5.2. Figure 15 graphically illustrates the behavior of the analyses. When the ratio r/d is 0.14 and 0.18 with convergence criteria of 5% and 2% presented the greatest difference. On the other hand, in these same points considering a convergence criterion of 1% the variation presented was 0.7% and -0.1%.

Figure 15 Stress concentration factor K with respect to the ratio r/d.



7. Conclusions

Based on the analysis of comparative results between a theoretical and numerical study carried out on a flat stepped plate, the average stress concentration factor varied 2.5%, 0.9% and 0.6% with a convergence criterion of 5%, 2% and 1%, respectively. This using the tool “Convergence”. Also, points were found where the difference of the variation is maximum, being for a convergence criterion of 5% a variation of 10.4%; for a convergence criterion of 2% a variation of 8.7%; and for a convergence criterion of 1% a variation of 3.7%.

The results of numerical studies are approximations that depend on the discretizing process. As a more rigorous convergence criterion is applied, the theoretical and numerical results are closer. Further refinement impacts the study execution time and computational resource capacity required. In the case of academic leave used, at one of the design points the permissible limit was reached.

8. Referencias

ANSYS. (2023). *Download Ansys Student | Workbench-based Simulation Tools*. https://www.ansys.com/academic/students/ansys-student

Budynas, R. G., & Sadegh, A. M. (2020). *Roark’s Formulas for Stress and Strain* (9th Edition). McGraw-Hill Education. https://www.accessengineeringlibrary.com/content/book/9781260453751

Chen, X., & Liu, Y. (2018). *Finite Element Modeling and Simulation with ANSYS Workbench* (2nd ed.). CRC Press. https://www.routledge.com/Finite-Element-Modeling-and-Simulation-with-ANSYS-Workbench-Second-Edition/Chen-Liu/p/book/9781138486294

Chmelko, V., Harakal’, M., Žlábek, P., Margetin, M., & Ďurka, R. (2021). Simulation of Stress Concentrations in Notches. *Metals 2022, Vol. 12, Page 43*, *12*(1), 43. https://doi.org/10.3390/MET12010043, https://www.mdpi.com/2075-4701/12/1/43.

Ferdinand, P. B., Johnston, E. R., DeWolf, J. T., & Mazurek, D. F. (2020). *Mechanics of Materials* (8th ed.). McGraw-Hill Education.

Goodno, B. J., & Gere, J. M. (2018). *Mechanics of Materials* (9th ed.). Cengage Learning.

R. C. Hibbeler. (2018). *Mechanics of Materials* (10th ed.). Pearson Education.

Roldan, F., & Bastidas, U. (2002). Estudio Experimental y por Análisis de Elementos Finitos del Factor de Concentrador de Esfuerzo Producido por un Agujero en una Placa Plana. *Dyna*, *69*(137), 1–8. https://www.redalyc.org/articulo.oa?id=49613701

Studies in Photoelastic Stress Determination | J. Fluids Eng. | ASME Digital Collection. (1934). https://asmedigitalcollection.asme.org/fluidsengineering/article-abstract/56/7/637/1157771/Studies-in-Photoelastic-Stress-Determination?redirectedFrom=fulltext

Wilson, I. H., & White, D. J. (1973). Stress-concentration factors for shoulder fillets and grooves in plates. *Journal of Strain Analysis*, *8*(1), 43–51. https://doi.org/10.1243/03093247V081043, https://journals.sagepub.com/doi/10.1243/03093247V081043