International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 4 (2021), pp. 339-346

Description of the meshing process in tool wear analysis

Jonathan Javier Pabon[1], Luz Karime Hernandez Gegen[2] and Sir-Alexci Suarez Castrillon[3]

[1,2] Faculty of Engineering and Architecture. University of Pamplona, Colombia.
[3] Engineering Faculty, University Francisco of Paula Santander Ocaña, Colombia.

Abstract

The present work describes the CAD development and pre-processing for finite element analysis of a rhombic monophilic cutting plate, used as a cutting tool in turning processes. For this purpose, the CAD of the mentioned geometry is subjected to different meshing and refinement procedures, varying the number of nodes or mesh density, before and after applying a cleaning method, at the same time the influence of the geometry cleaning on the meshing continuity is determined.

Keywords: CAD, geometry cleaning, meshing, nodes.

1. INTRODUCTION

In engineering analysis, it is common to use computer-aided design (CAD), whose process consists of developing models with the aid of a computer. This methodology allows the modeling and analysis of three-dimensional (3D) objects [1], expresses in his work, "the creation of quality images in CAD programs depends on Render", clarifying that, to achieve a realistic form of light, shadows, reflections and textures CAD graphics programs make intense mathematical calculations that apply the laws of physics. Thanks to the use of the computer in design, the creation of figures and geometries has been improving to the point of being able to see them before they are created. Similarly, these shapes can be analyzed by means of computer-aided engineering (CAE), which with the same computational aid provides insight into the physical and mechanical behavior. The analysis represents a computational expense, which is why it is important to define parameters and conditions for the simulation. The parameter studied in this work is the relationship between mesh density and quality of the computational analysis of the mesh. It is important the cleanliness of the geometry and an optimal meshing since it is directly related to the computational cost.

Similar studies seek to use finite element modeling to evaluate cutting tool wear, who propose a CAE model with the objective of simulating this tool deterioration and heat production. The evaluated results revealed that the model could predict the tool flank wear rate with acceptable errors. Similarly [2], wanted to evaluate the impact of drilling nickel base alloys (Inconel 718) and developed and implemented a CAE wear model that obtained an acceptable similarity between the predicted and measured data. On the other hand, Kumar et al [3], performed a study where they modeled a femur bone in 3D with the help of Solid Edge software, to be later analyzed in a finite element model (FEM) with ANSYS software. The study was based on stress analysis to analyze contact stresses in the hip when performing activities such as walking, running, jumping and standing. It was compared with experimental results to optimize the femur design.

As can be seen, computational advances provide tools for assisted drawing. By means of which it is possible to design or model any element that the imagination or reality wants to achieve. From the CAD it is possible to analyze such structure and make it a reality through manufacturing.

Different researchers have ventured into the study of meshing geometries for an efficient computational analysis, as is the case of Ali et al [4]. This team carried out a study of meshes for flow exercises in CFD where they analyze the most used meshing techniques, with the objective of reducing the processing time. The researchers considered certain automatic meshing strategies, which they evaluated with an error analysis mechanism. They determined that, with methods based on medial axis, a better flow alignment and more uniform cell size distribution is produced. On the other hand, in order to develop new models to evaluate the mesh stiffness of a gear mesh in a time-varying dynamic analysis, Liang et al [5] proposed three models in their study on gear dynamics. They used finite element analysis in order to evaluate the mentioned elements, obtaining a low error in two of the three models. In a similar study, examined a CAE method to find a diagnosis in gear failure [6]. They relied on the stiffness of the mesh by studying the crack propagation path as these reflect the severity of tooth damage. The study of meshing is necessary in different applications. Without leaving aside a topic of worldwide influence such as renewable energy. Two investigations are presented below. Advis et al. [7] with the aim of preventing hydrodynamic changes affecting the local and regional ecology in the installation of a tidal turbine farm in the Orkney Islands. They performed a robust mesh generation model to develop high accuracy simulations. They tested their predictions with two well-validated ocean models. On the other hand, Espinosa et al. [8] Evaluated tubular K-joints of wind turbines by doing a meshing influence study. Several meshing designs are considered for the hot spot method using numerical methods. Two meshing models, semi-automatic and automatic, were tested to compare their effect on the simulations. They conclude that the use of automatically generated mesh is justified in certain cases.

In this project, we intend to model a tool (insert) used for the turning process. The modeling starts with the development of the CAD in Solid Edge software. Subsequently, this design is subjected to a geometry cleaning process which is carried out with ANSYS software, leaving the part ready to perform the wear analysis of the same. This article shows the CAD modeling, the selection of the appropriate mesh type for further processing and a comparison of meshing methods in relation to the number of nodes. The parameter to evaluate the quality of the geometry is going to be the mesh density. The objective is to obtain a meshing that generates a lower computational cost in a simulation analysis process.

2. METHODOLOGY

For a subsequent analysis of a part or set of parts by means of simulation, it was necessary to design the solid to be studied. This was developed using computational tools that provide the ability to model and have a 3D vision adjusted to the real geometry. The construction of the design implied knowing the part in its totality, that is to say, having all its geometric details defined. In order to have these details it is essential to specify the part to be analyzed as the object of study, in this case an insert or cutting insert used as a tool for machining processes in lathe operations. In order to fulfill one of the objectives of this work, a CAD is made to a monofilament tool (Figure 1 and 2) with its top and side views. It is important to know the reason for this selection, also the corresponding geometrical analysis and finally the development of the CAD. This process is listed below.

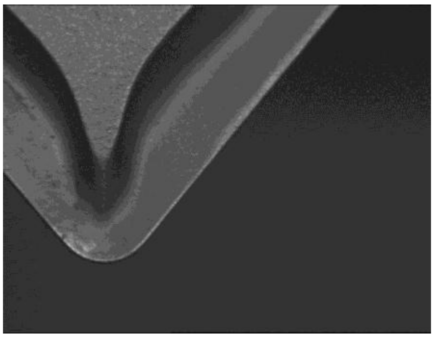


Fig. 1. Top view of monofilament cutting tool [9].

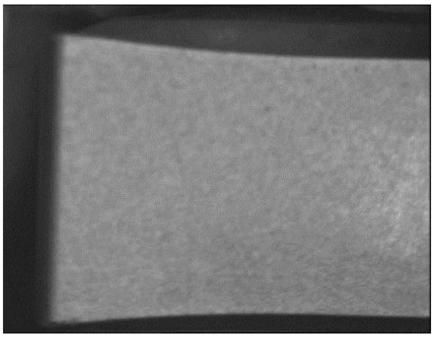


Fig. 2. Side view of monofilament cutting tool [9].

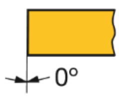
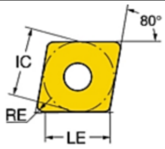
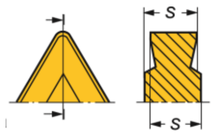
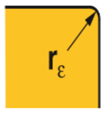
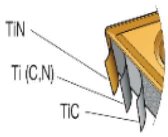
2.1 Insert selection

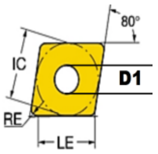
In this selection it was necessary to resort to the type of machining to be performed. To begin with, it was necessary to point out that the manufacturing process selected is turning and the type of machining is external turning. Likewise, the insert is distinguished by its codification as CN MG 12 04 04-MF 235 according to ISO code for inserts.

2.2 Geometric analysis of the insert

Once the insert with ISO code CN MG 12 04 04-MF 235 has been established, the shape and geometrical details important for the elaboration of the CAD are compiled. The geometry of this insert can be determined using the codes and information in Table 1. Data obtained according to the ISO code for insertsThe individual components of this code determine a part or shape of the geometry, so it is important to collect all this information.

Table 1. Data obtained according to the ISO code for inserts [10].

Code.	#	Equivalence	Data obtained
C	1	Form	 Rhombic with 80° angle
N		Relief angle	 Angle of 0° degrees
M		Tolerance	S= +- 0.013 mm iC= +- 0.013
G		Chip breaking and clamping	 4 edges and shape of the chip breaker
	5	Edge length	 IC= 12.7 mm
		Thickness	S= 4.7625mm 
		Radius size	 RE= 0.4 mm RE=0.3969mm
MF		Chip breaker	Designated by the manufacturer.
235		Grade	 Quality of the insert designated by the manufacturer.

-	-	Other measures	 LE=12.4959mm L= 12.8959 mm D1=5,156mm Weight WT= 0.01kg

Similarly, Fig. 1shows each data obtained and the corresponding geometric application [11].

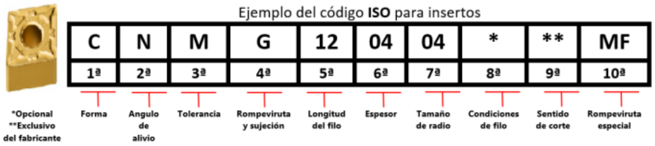


Fig. 12. ISO code for inserts with equivalence [11]

In addition to this data, it is necessary to know the basic composition of the material to be machined, hardness and machinability. This is known with the designation of a letter and distinctive color as shown in Fig. 3It also provides ideal cutting parameters, the blue color indicates that this is an insert for machining carbon steel while yellow indicates its usefulness for stainless steels.

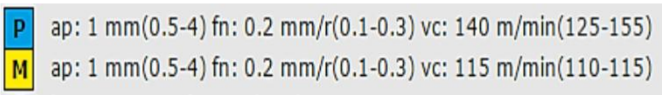


Fig. 34. Insert composition [11].

The geometric details obtained so far are a fundamental basis for the correct development of the CAD. However, they are not sufficient when modeling more specific parts such as the chip breaker, the clamping system, angles and curvatures. These geometric data are obtained by taking manual measurements. As shown in Fig. **5**.

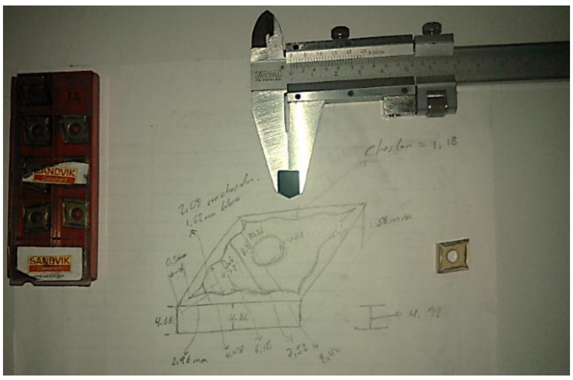


Fig. 56. Sketch of manual measurements of the insert.

2.3 CAD development

Once the insert has been defined and all its geometry has been determined, the respective design is developed using computer tools. For the development of the CAD, there is no specific order of operations. This depends on the decision of the software operator. However, the following elementary operations were defined in its progress.

A. Sketch and general structure of the insert

In the original sketch the surface geometric measurements are taken into account along with the angles at each of the tips of the insert and the circle inscribed in the parallelepiped formed Figure 6 (a). Followed by the extrusion in an isometric plane of the solid in 3D where there are curvature cuts that end at the edge of the tool.

Error! Reference source not found.(b) shows this first operation of the CAD elaboration process, with the respective dimensions using the measurements taken directly from the plate.

B. Internal edge chamfering and edge rounding

The importance of the chamfer lies in the correct chip evacuation, this implies to be able to direct and have control of it, directly to break the chip. For its design it is necessary to take physical measurements of the insert. The rounding data on the cutting edges are obtained from the manufacturer and depend on the process to be performed including the material and surface finish. These operations are shown in (c and d). 6(c and d).

C. Chipbreaker

The chip breaker is designated by the manufacturer with reference "MF" (code established by the manufacturer). The chipbreakers vary in the angle of inclination depending on the hardness of the machining as well as the required surface finish. The MF reference allows the turning process of materials with high hardness and is also important for the tool life. In this process, the entire geometry of the chip breaker was measured to the insert with the help of the vernier caliper, to then continue with the design. **Error! Reference source not found.**6 (e) shows the geometry of the chip breaker as its sketch.

D. Fastening hole

The fastening system is arranged by means of a hole in which, with the help of a bolt, the insert is securely mounted. For the development of the hole, only the manufacturer's dimensions and tolerances must be followed. This hole is important together with its tolerance to avoid vibrations and a good handling of the tool. See **Error! Reference source not found.**6 (f).

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 4 (2021), pp. 339-346

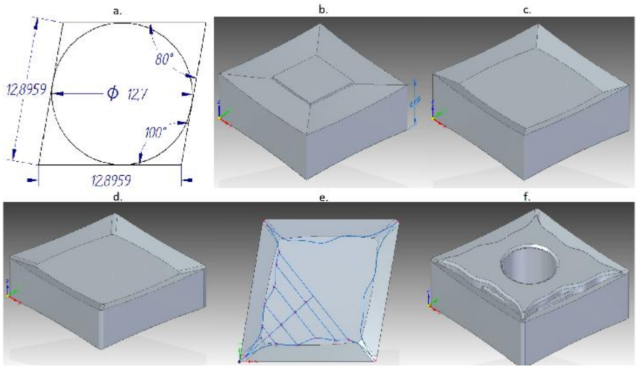


Fig. 6. Development of the CAD.

The finish of the insert is gold colored (gold in the CAD color bar) due to its TiN titanium nitride coating as presented in Fig. **7**

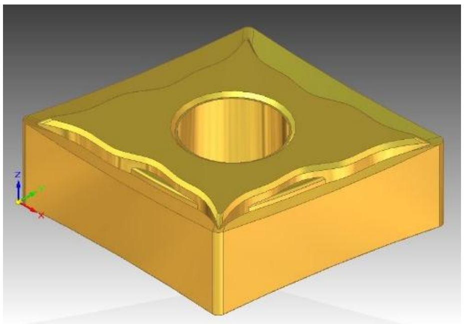


Fig. 7. Tool finishing

E. Map

Fig. **8**8 shows the drawing with the front, right lateral, top and isometric views and a detailed enlargement of the cutting tip, in order to observe the characteristics of the cutting side of the tool. These views show the measurements used for the construction of the CAD and each of its minor parts.

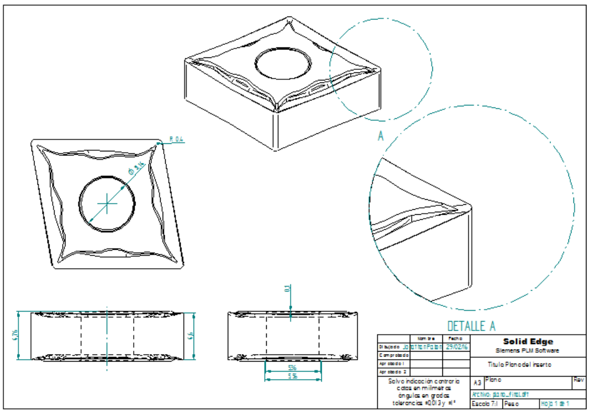
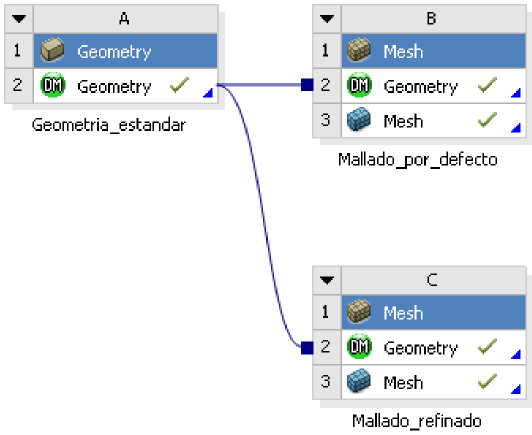


Fig. 8. Insert plan.

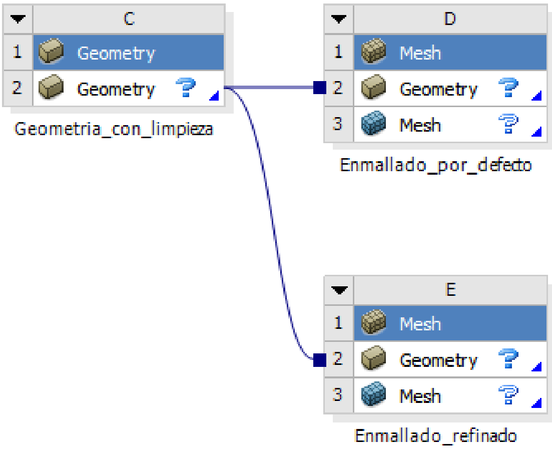
2.4 Geometry Cleaning and Meshing

In a finite element analysis, the meshing process consumes a large percentage of time. In this process, the parts of interest were selected where mesh refinement was applied in order to obtain accurate results in a particular area. However, it is important to keep in mind that the number of nodes can be increased infinitely. Due to the computational expense involved in having an excess mesh, it is correct to define a refinement that meets the conditions of the exercise and manages a ratio of accuracy of results. Several parameters interfere in the efficiency of a meshing, one of these is the simplicity of the geometry to be analyzed. A geometry with abrupt changes in its surface involves a more detailed meshing. In this work we used a geometry correction tool where not only import defects are corrected, but also areas with sharp changes in the geometry are modified. The development of this phase of the work is presented below.

In this process the geometry was imported twice into ANSYS software, the first geometry underwent a normal meshing process and the second one underwent a geometry cleaning process before meshing. Fig. **9**9 shows the import and modeling of the two geometries in the software.



a. Meshing process with standard geometry

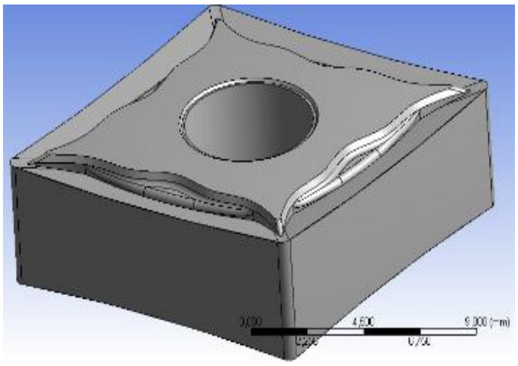


b. Meshing process with geometry cleaning

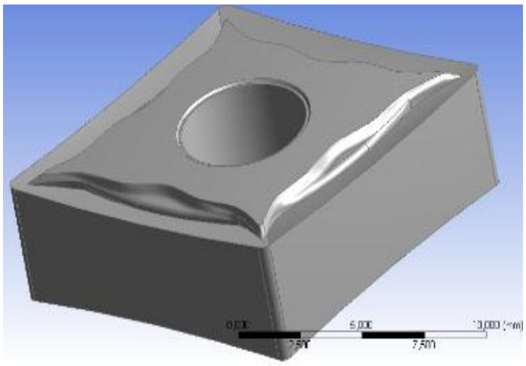
Fig. 9. Working model in ANSYS software.

A. Geometry cleaning

In the cleaning process, ANSYS software tools were applied, which analyze the geometry and detect errors in lines, angles, surfaces, holes, among others. This was carried out in order and determining geometric details that require repair. Fig. 10 shows the before and after cleaning process.



a. Original figure without cleaning



b. repair process.

Fig. 10. Cleaning process.

In the cleaning process, the repair tool of ANSYS software was applied, which displays several options for the repair of hard edges, joints, holes, acute angles, splinters, peaks, bodges, faces and bodies. In this cleaning process, first the face repair was carried out with a total of 50 corrections. This was followed by the repair of 50 acute angles, 50 faces until finishing with a total of 28 chips.

Fig. 11 shows the edge repair process, where the green dots are the sections accepted for repair and the black dots are the vertices present.

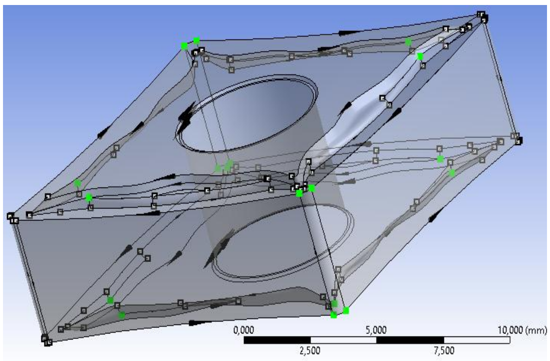


Fig. 11. Edge repair process

B. Mallado

By means of the software the meshing process is carried out, there is a default method in which the software takes the part for a general analysis. In the case of the insert, it is necessary to know in more detail the effects of the forces and deformations acting on a certain face or part of the geometry. ANSYS allows to modify the mesh density at a certain point of the geometry, as well as other tools for specific cases. For this study, the analysis is focused on the tool tip. Therefore, a refinement of the mesh is performed on the edge that performs the cutting function, as it is the only one that comes into contact at the time of machining.

Standard mesh

The standard meshing process is performed in the two geometries as shown in Fig12. In the geometry on the left, the standard meshing performed by the software by default can be seen, while on the right, the insert is meshed with its corrected geometry.

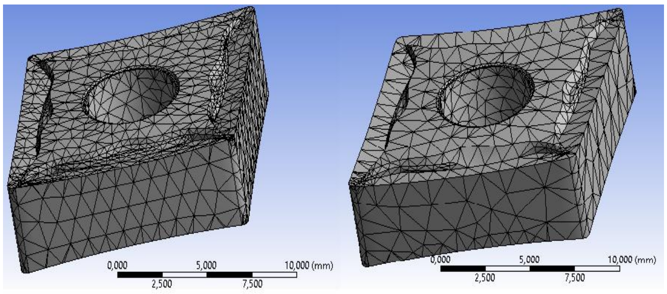


Fig. 12. Standard meshing figure on the left without cleaning, on the right with the repair process.

Meshing with refinement

In the mesh refinement process, the Refinement tool of the software was used, selecting a face and edge that are of direct incidence in the turning process. In Fig13 you can see and compare the refinement performed.

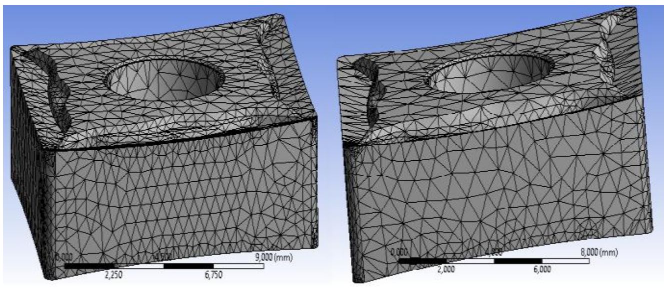


Fig. 13. Meshing with refinement figure on the left without cleaning, on the right with the repair process.

However, in spite of the refinement, it can be observed that the mesh is not orderly and the number of nodes in the whole figure is high. When performing the simulation, the number of nodes is a relevant factor for both the accuracy of the results and the computational cost, so it is possible to opt for a less dense mesh and the refinement more focused on the area of interest. In the example in Fig14, an element size of 2 mm and a refinement in grade 3 only in the contact tip of the tool were used, obtaining a total of 18308 nodes in the geometry without cleaning and 23579 nodes in the geometry with cleaning.

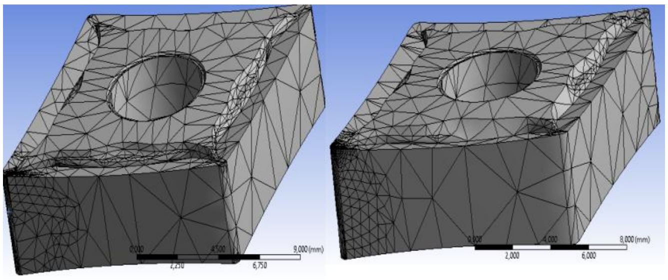


Fig. 14. Size control and refinement figure on the left without cleaning, on the right with the repair process.

Although the refinement in the zone of interest is good and can provide reliable information when simulating, the number of nodes is still high. For this reason, a cut section was taken leaving only the tool tip as the meshing and analysis part. This allows to control the mesh size and to emphasize only the point of interest without taking into account the rest of the part. This is due to the fact that the only area affected at the moment of the tool-part contact is the tip in question. Fig15 shows the final model to be taken to the simulation and the type of meshing that was performed obtaining a total of 10898 nodes in the geometry with the repair process.

However, for an initial analysis, a simplified geometry can be taken into account, as in the case of

Fig**. 15.** Final model using a cut section of the area of interest.

16, where not only a cut of the tip is performed, but also a geometry without edges or changes, as is the case of the chip breaker that alters the meshing process. In this figure a total number of 4268 nodes is presented, perfect for an initial experimental analysis that involves less processing time, allowing to modify parameters in a later finite element simulation.

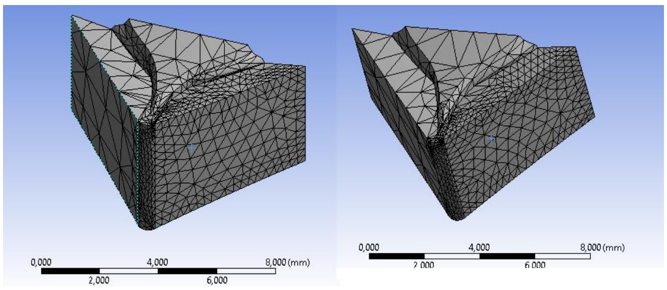


Fig. 15. Final model using a cut section of the area of interest.

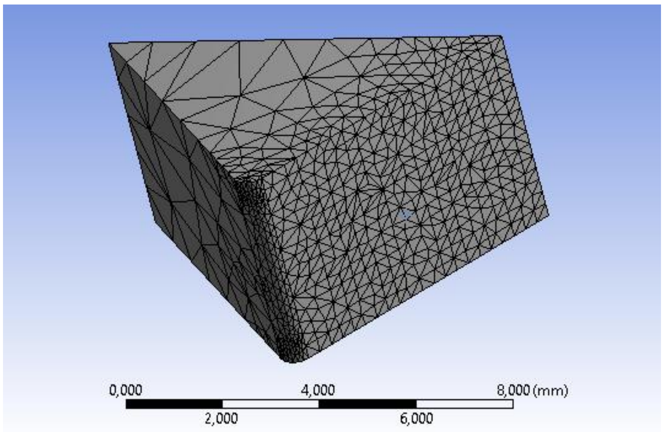


Fig. 16. Model with simplified geometry.

3. RESULTS

Geometry cleaning

After the CAD cleaning process, a geometry with less sharp edges, close to the real insert, is obtained. The cleaning process eliminates geometric details arising from the import process, such as errors in line connections, holes, angles and faces. Eliminating sharp edges from the geometry not only implies less meshing details in unimportant areas, but also a tidying of the mesh to prioritize the analysis in the area of interest.

Mallado

In the mesh analysis, not only the number of nodes and elements can be observed, but also their arrangement. A mesh with appropriate and ordered number of nodes provides analysis results with less processing time. Table 2summarizes the total number of nodes and elements obtained after each process.

Table 23. Number of nodes and elements in each mesh.

	Standard geometry		Geometry with cleanliness	
	# of nodes	# of elements	# of nodes	# of elements
Default meshing	20545	11821	13201	7665
Refined mesh	28360	16821	21530	13034
Size control and refinement	18308	11247	23579	15443
Final geometry tip	12830	8280	10898	7013

In the two types of meshing, the default and the refined meshing, a smaller number of nodes and elements in the geometry subjected to the cleaning process is observed, which results in computational savings.

Fig17 shows by means of a bar diagram the number of nodes required for each of the geometries according to the type of meshing and refinement. It includes the final geometry which is only a cut section of the original one and in which a total of 12830 nodes were obtained in the geometry without cleaning and 10898 nodes in the geometry with the repair process.

At the same time, a mesh with better ordering in the standard geometry can be observed, regardless of the type of meshing. Taking into account the computational expense, it could be said (depending on the exercise to be simulated), that a smaller number of nodes implies less processing, so we try to reduce the number of nodes without sacrificing the quality of the results to be obtained. For this reason, the geometry selected for the subsequent simulation will be the one that involves only the section of interest.

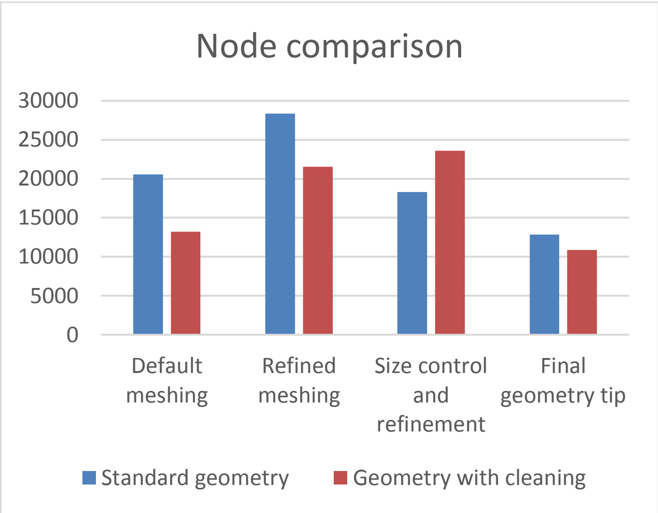


Fig. 17. Comparison of the number of nodes in the cases analyzed.

4. CONCLUSION

A significant change in the geometry is observed once the cleaning process is performed. When compared to the standard geometry, the number of nodes is reduced by approximately 35% in the application of a default mesh. Likewise, in a refined meshing it is reduced by approximately 20%.

A significant change in the geometry of the tool is observed once the cleaning process is performed. When compared with the standard geometry and the clean one, the number of nodes presents a reduction of 35% in a default mesh; if compared with a refined mesh, the reduction of nodes is 20% after cleaning.

In order to save computational cost, it is relevant to identify the critical area of analysis in the geometry, which requires a refinement of the same and pay special attention only to this area. This study shows that by simplifying the geometry and minimizing the study area, the mesh density can be reduced by up to 80%.

It is important to clean the geometry when the objective of the geometry is a simulation or before a finite element analysis. This cleaning involves the possibility of prioritizing in a specific area.

REFERENCES

[1] P. Nicoleta-Elisabeta, T. Dobrescu, C. Opran, y G. Enciu, «Realistic Scenes in CAD Application», *Procedia Engineering*, vol. 69, pp. 304-309, dic. 2014, doi: 10.1016/j.proeng.2014.02.236.

[2] H.-F. Wang y Y.-L. Zhang, «CAD/CAM integrated system in collaborative development environment», *Robotics and Computer-Integrated Manufacturing*, vol. 18, n.º 2, pp. 135-145, abr. 2002, doi: 10.1016/S0736-5845(01)00027-8.

[3] K. C. N. Kumar, T. Tandon, P. Silori, y A. Shaikh, «Biomechanical Stress Analysis of a Human Femur Bone Using ANSYS», *Materials Today: Proceedings*, vol. 2, n.º 4, pp. 2115-2120, ene. 2015, doi: 10.1016/j.matpr.2015.07.211.

[4] Z. Ali, P. G. Tucker, y S. Shahpar, «Optimal mesh topology generation for CFD», *Computer Methods in Applied Mechanics and Engineering*, vol. 317, pp. 431-457, abr. 2017, doi: 10.1016/j.cma.2016.12.001.

[5] X. Liang, H. Zhang, M. J. Zuo, y Y. Qin, «Three new models for evaluation of standard involute spur gear mesh stiffness», *Mechanical Systems and Signal Processing*, vol. 101, pp. 424-434, feb. 2018, doi: 10.1016/j.ymssp.2017.09.005.

[6] C. P. Rivera-Guerrero, E. G. Aguirre-Parra, J. Medrano-Bautista, y P. Rojas-Gomez, «Tecnología CAD/CAM en la consulta dental», *Dominio de las Ciencias*, vol. 3, n.º 2, Art. n.º 2, mar. 2017, doi: 10.23857/dc.v3i2.356.

[7] A. Avdis, A. S. Candy, J. Hill, S. C. Kramer, y M. D. Piggott, «Efficient unstructured mesh generation for marine renewable energy applications», *Renewable Energy*, vol. 116, pp. 842-856, feb. 2018, doi: 10.1016/j.renene.2017.09.058.

[8] J. M. Espinosa, S. Schafhirt, y M. Muskulus, «Mesh Parametric Study for Fatigue Assessment of Tubular K-joints using Numerical Methods», *Energy Procedia*, vol. 137, pp. 514-522, oct. 2017, doi: 10.1016/j.egypro.2017.10.380.

[9] L. K. H. Gegen, «Propuesta de nuevos criterios de vida de herramienta de corte y determinación del desgaste a partir de la clasificación de imágenes digitales caracterizadas utilizando descriptores de región simples y basados en momentos», *Ciudad de León*, 2007.

[10] Q. Martínez, «Mecanizado de piezas pequeñas - Umesal Mecanizados», *Umesal*, may 12, 2018. https://umesal.com/mecanizado-de-piezas-pequenas/ (accedido may 11, 2021).

[11] C. M. Villar, «Selección y Consejos sobre Insertos Intercambiables Para Torneado», *Metal actual*, 2010.