International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 12, December-2014

Simulation using a Numerical Methods in a Deep Fry Basket to Obtain the Mechanical Stress Analysis

Aurora Yamilet Hernández Hernández
Universidad Autónoma de Nuevo León, Departamento de Mecátronica, Facultad de Ingeniería Mecánica y Eléctrica, Nuevo León, México.

Guadalupe Maribel Hernández Muñoz
Universidad Autónoma de Nuevo León, Posgrado de Ingeniería con Orientación en Manufactura, Facultad de Ingeniería Mecánica y Eléctrica, Nuevo León, México.

Nelda Johana Gámez Treviño
Universidad Autónoma de Nuevo León, Posgrado de Ingeniería con Orientación en Manufactura, Facultad de Ingeniería Mecánica y Eléctrica, Nuevo León, México.

Flor Araceli Garcia Castillo
Universidad Autónoma de Nuevo León, Departamento de Sistemas Integrados de Manufactura, Facultad de Ingeniería Mecánica y Eléctrica, Nuevo León, México.

***Abstract*— This work shows a stress analysis in a Deep Fry Basket (DFB) taking care in each mechanical properties and the operating conditions using numerical methods to obtain this results. The design of this component has to consider supporting high temperatures and different loads. Actually it is necessary to provide information of de design using Computer Assistance Design (CAD) programs to assure the new products are ready for immediate production upon sample approval.**
Also this period is characterized by the discovery of new materials offering technological possibilities only dreamed, by the development of these materials is possible only when they are very special properties testing new materials is an option to increases its shelf life, the design using computer tools (Computer Assistance Design) and simulation using Computer Assistance Engineering (CAE) helps to analyze the mechanical behavior and select the material that meets operational specifications that will be exposed.
The article presents a detailed comparison of the material used and one proposed by analyzing the mechanical stress to select the best.

Keywords— Mechanical Stress, CAE, CAD

I. INTRODUCTION

The manufacture of new products considers every day more complex models, the develop of new materials more resistant and with different operating conditions leads us to develop techniques process of modeling and simulation to predict their behavior and to choose the design, or the material and most suitable operating conditions to ensure the users satisfaction.

There are different modeling CAD (Computer Assistance Design) programs which are faster and more accurate than manual design techniques and also with tools to analyze mechanical behaviors, CAE (Computer Engineering Assistance) reduces product development time and improved product quality and durability, these tools become an inherent part of the development process of the present product [1].

A variety of commercial CAD software as Solidworks, Pro / Engineer®, Abacus® and computational analysis tools like Ansys CAE® and NX NastranTM® are widely used in industry, in this work the computer software use is named Solidworks® this is because we can model the component and also we can make a finite analysis for the mechanical behavior of a Deep Fry Basket (DFB), this is designed to use in homes for the kitchen. The design has a basket to keep the large and small particles contained and thanks to its element coated handle will also be reducing your chances of receiving a nasty burn. New materials are propose and can withstand high temperature and different loads and meet the specifications to which this component will be exposed [2].
In this paper numerical methods are utilized to arrive an approximate solution of claim to see the results in the analysis of design of the DFB, and select the material and its optimum operating conditions.

II. MECHANICAL PROPERTIES

A variety DFB, are used for frying different foods in the kitchen, previously this component was composed of low carbide steel, this material is used because of their temperature resistance and other mechanical requirements. Carbon steel is steel in which the main interstitial alloying constituent is carbon in the range of 0.12–2.0%. The American Iron and Steel Institute (AISI) defines carbon steel as the following: "Steel is considered to be carbon steel when no minimum content is specified or required for chromium, cobalt, molybdenum, nickel, niobium, titanium, tungsten, vanadium or zirconium, or any other element to be added to obtain a desired alloying effect; when the specified minimum for copper does not exceed 0.40 percent; or when the maximum content specified for any of the following elements does not exceed the percentages noted: manganese 1.65, silicon 0.60, copper 0.60 [3]. These steels have a low cost, but have some limitations; they have little resistance to corrosion [4].

In this work several materials testing first is a 1020 low carbon steel (AISI 1020) which must meet the following mechanical properties shown in Table 1.

TABLE I: MECHANICAL PROPERTIES OF AISI 1020 [5].

Maximum effort (σmax)	552 Mpa
Effort creep (σ)	345 Mpa
Elongation 2 en 2%	20

Currently DFB, must have greater resistance of changes whereby materials can withstand the high temperatures that will be exposed this component, in the second simulation of this work a stainless steel alloy is propose, the advantage are the high corrosion resistance because of the high chromium content. Stainless steel is used where both the mechanical properties of steel and corrosion resistance are required. This 446 stainless steel alloy has to comply with the following mechanical properties shown in Table 2.

TABLE II: MECHANICAL PROPERTIES OF STAINLESS STEEL 446 [5].

Maximum effort (σmax)	552 Mpa
Effort creep (σ)	345 Mpa
Elongation 2 en 2%	20

For both materials we considered the Poisson ratio of 0.291.

III. NUMERICAL METHODS

The finite element method (FEM) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. This method use variational methods (the calculus of variations) to minimize an error function and produce a stable solution. Analogous to the idea that connecting many tiny straight lines can approximate a larger circle, FEM encompasses all the methods for connecting many simple element equations over many small subdomains, named finite elements, to approximate a more complex equation over a larger domain.

The solution approach is based either on eliminating the steady state problems represented by discrete equations, or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard mathematic techniques such as Euler's method, Runge-Kutta, etc. [6].

FEM used a more general approach to the method of finite differences or finite volume method, by using a larger number of function spaces this generality makes it suitable for use with complex, such as cars and pipelines, motion systems, or complex process like wax melting domains. [7].

There are several computer programs capable to solve complex numerical models and obtain approximate solutions of their behavior actually it is vital for the development of new products and the selection of materials with improved mechanical properties the application of these tools.

IV. EXPERIMENTAL

This study help to find techniques to avoid unexpected fractures and to ensure the safe operation of a DFB in the kitchen, a maximum load is applied at the bottom line to check the behavior of the material and see that the solder points between basket and metal arm will not break under the applied load. This paper also make a comparison of the material used and one proposed to select the best behavior

A. Geometry

The modeling process took 3 steps that we mention next. First we model the component using Solidworks®. To build a master model of the Basket handle shown in Figure 1 using a sketch with primitive shapes such as arcs, lines and ellipse, then we converted to a solid part using the sweep command. The upper and the lower parts from the Master model shown in Figure 2 making a cut plane, and doing a shell process. To make the solid hollow we considered 2mm as thickness to have a shell structure.

Fig.1. MODEL OF THE BASKET HANDLE

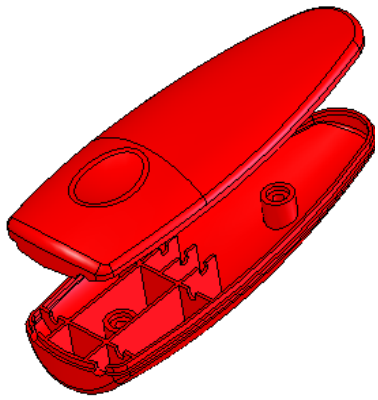

Fig. 2. THE UPPER & THE LOWER PARTS FROM THE MODEL EASURES BASKET

The second step was Build the metal arm and Building the basket using a rectangle with the center at the origin and considering the dimensions of 6.5 in x 51/8 in, then we used an extrusion to model the 3D shape, the fillet on surface edges .59 in of radius, to add material texture as shown in Figure 4, we select from the menu of the textures.

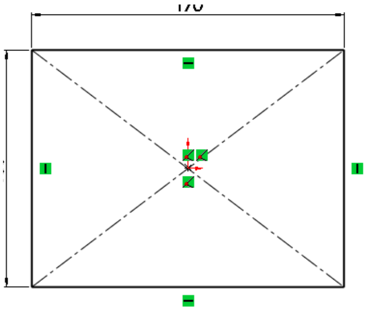

Fig.3. MEASURES BASKET

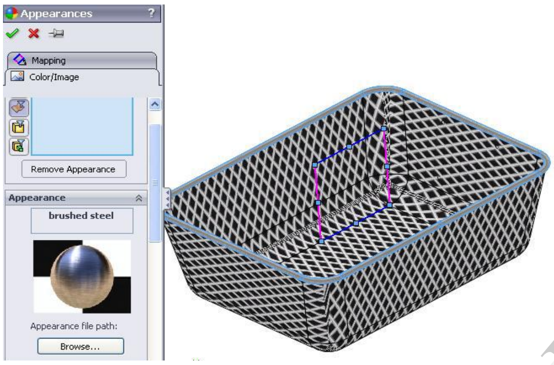

Fig. 4. MESH TEXTURE

The arm of the basket was modeled using spilines giving them the x, y and z coordinates and using a mirror command to approach the symmetry.

The third step was the assemble of the components together, considering all the components handle, arm and the basket using all the mate conditions such as coincident points, concentric holes and design distances. The complete component is shown in figure 6.

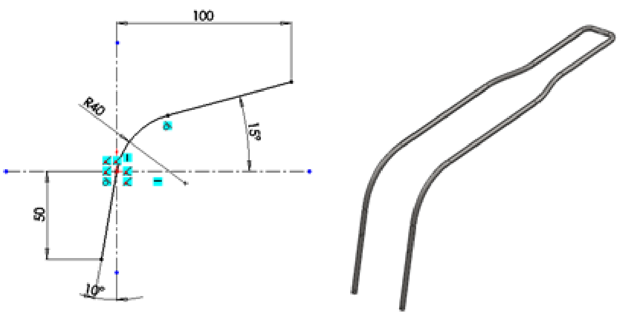

Fig.5. DIMENSIONAL OF METAL ARM

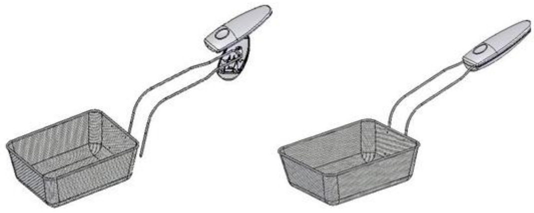

Fig. 6. COMPLETE COMPONENT

B. *Simulation*

The simulation was a static analysis using a numerical solution. The mesh for all the components was of 0.2in as global size, the restriction conditions were created and a fixed geometry is selected, a distributed force of the basket as shown in the next figure with a total area value of 20238mm2 obtained by the software. The next equation gives us the total pressure

$$P=F/A \qquad (1)$$

$$P=15N/202380e^{6}m$$
$$P=741.2\ N/m^{2}$$

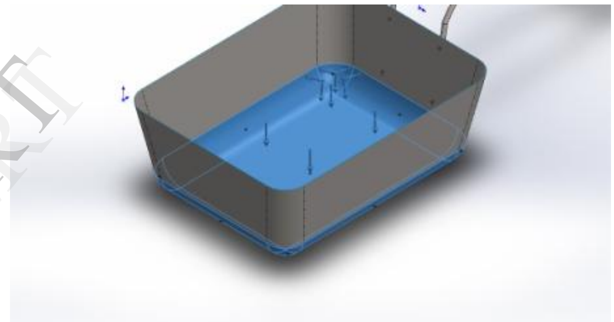

Fig.7. DISTRIBUTED EXTERNAL PRESSURE LOADS

The types of materials used in the analysis were defined with the mechanical properties describe in point 2; this work shows the displacement and strain that has the basket.

V. RESULTS

Figure 8 describe the von Mises stress distribution (in Pascals) for the first simulation (AISI 1020) and a maximum von Misses stress of 1, 134.67 MPa was found between the solder points between basket and metal arm (marked with red circles) showing the possible failure area.

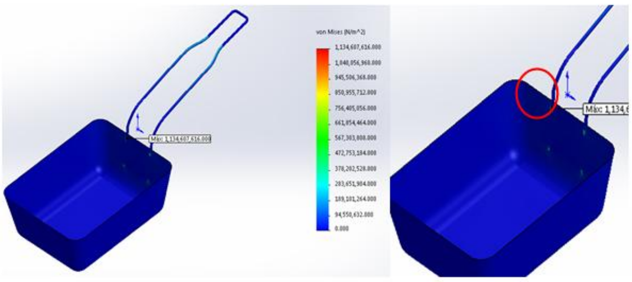

Fig. 8. BASKET RESULTS GIVEN TENSION FOR AISI 1020

The figure 9 also shows the von Mises stress distribution (in Pascals) for the second simulation (stainless steel) the maximum stress was of 1, 134.67 MPa in the solder points between basket and metal arm (marked with red circles).

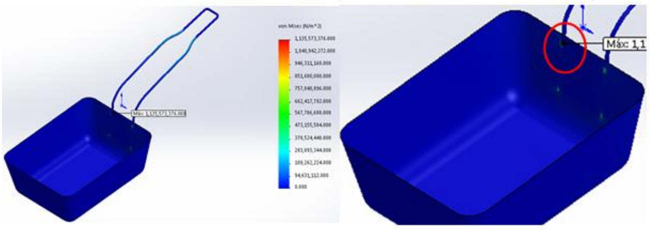

Fig. 8. BASKET RESULTS GIVEN TENSION FOR STAINLESS STEEL

VI. CONCLUSIONS

This study has shown an alternative to analyze the structural behavior of these components. The results of stresses in the static simulation was almost the same for both materials, therefore the design of these containers it is suggested to consider different materials with different mechanical properties to be able to support operational and loading design conditions. Also is necessary to propose a different design to see the behavior in the solder points.

More work has to be carried out in order to obtain the thermal behavior. The heat transfer coefficient must depend on the temperature, and it needs not to be taken as a constant value. Cycles of mechanical and thermal loading need to be taken into account so a fatigue analysis is also necessary get more realistic results.

ACKNOWLEDGMENT

The authors acknowledge support from Universidad Autónoma de Nuevo León (México) by the concession of a fellowship PROMEP.

REFERENCES

[1] Siemens;http://www.plm.automation.siemens.com/en_us/plm/cae.shtml
[2] G.P. Gujarathi. (2011). Parametric CAD/CAE integration using a common data model. Journal of Manufacturing Systems, pp 112-132.
[3] AISI Classification of Carbon and Low-Alloy Steels
[4] Francois Cardarelli. (2008). Springer-Verlag, Ferrous Metals and Their Alloys, pp 59-157
[5] William F. Smith. (1993). Fundamentos de la ciencia e ingeniería de materiales. Mc Graw Hill, Second edition, pp 534.
[6] J. Gamez Treviño, Manuel R. Piña Monarez, Jesús H. Garcia1 and Martha P. Guerrero Mata. (2009). A Stress Predicting Model For Deep Water Pipes Base don 3D Fem Analysis. International Journal Industrial Engineering, Special Issue, pp-436
[7] (1980). International Journal for Numerical Methods in Engineering, Vol. 15, pp 1207–1220
[8] Norambuena-Contreras, J. Castro-Fresno, D. Del Coz, J.J.García, P.J. (2011). Numerical simulation asphalt of experiment mixture by FEM and design, Vol. 10, pp 4-15
[9] G. Dieter. (1986 April). Mechanical Metallurgy. McGraw Hill
[10] Suhas V. and Patankar. (1980). Numerical Heat Transfer and Fluid Flow. Taylor and Francis.
[11] Hibbit, "Abaqus/Explicit User's Manual", version 6.9-2.