

The Analysis of Duct Model Through Structural and Dynamic Schemes

S. H. Yahaya, J. M. Ali and M. R. Arham

Abstract—This paper presents the analysis of duct design using static and dynamic approaches. The static approach is used to find out applicability between the design and material applied. The material used in this paper is Thermoplastic Olefins (TPO). For the dynamic approach, the focusing is only on the CFD simulations. The fatigue life in this design and material applied also covered.

Keywords—CFD, structural analysis, fluid analysis, duct design, fatigue life

I. INTRODUCTION

RECENT years have seen the transition of many industrial designs from drafting and prototyping the design manually to editing and tuning design via Computer-Aided Design (CAD)/ Computer-Aided Engineering (CAE)/ Computer-Aided Manufacturing (CAM) software. Now, more than ever, related industries are relying heavily on computational fluid dynamics (CFD) software as a tool to compute and visualize aerodynamics and as a replacement to experimental tests and designs. These CFD softwares are capable of simulating and modeling various physical phenomena virtually and possibly accurately.

CFD also referred as the numerical fluid mechanics [1], is one of the elements of fluid mechanics used in numerical methods [2]. For the presented study, ANSYS® CFX® package is used. This package consists of CFX-Pre, CFX-Solver and CFX-Post where all the physical properties such as material properties and boundary conditions as well as solver settings are set in CFX-Pre. Post-processing results are done in CFX-Post. The rationale for using CFX package are due to its capabilities of handling multiphase flows of any number of different fluids, all material properties can either be constant or dependent on any variable in the simulation, all problems can be calculated on structured and unstructured meshes, and most importantly CFX does not use a segregated solver for flow speed and pressure, but a fully coupled solver. In other words, CFX can perform all types of simulations on multiple processors [3]. The main focus in this paper is the design of duct that corresponding the good flow streamline and safe to be used. In this investigation, the application of CFD is

applied in duct design by presenting the end-to-end workflow and simulating the fluid motion. The duct geometry creation, meshing generation and structural analysis are all performed using CATIA V5 software.

This study examines an automobile air conditioning duct as an individual unit and not in the form of integration with the automobile instrument panel (dashboard) unit for the design by proposing the simple model.

II. METHODOLOGY

The overall process comprises four main components namely as Geometrical Design, Finite Element Modeling, Structural Analysis and Fluid Analysis as shown in Fig. 1. Geometrical Design is the CAD part of work, whereby the wireframe of the solid structure of the duct is design via any commercial CAD software. After completing the design, we can mathematically model the duct geometry using grids, also known as meshing. By using CATIA's OCTREE Tetrahedron Mesher, the duct is mesh automatically as unstructured mesh by defining the global mesh size equivalents to 7 mm, type of element is set to be linear and limit of geometry size (edge criterion) is set to 0.01 mm.

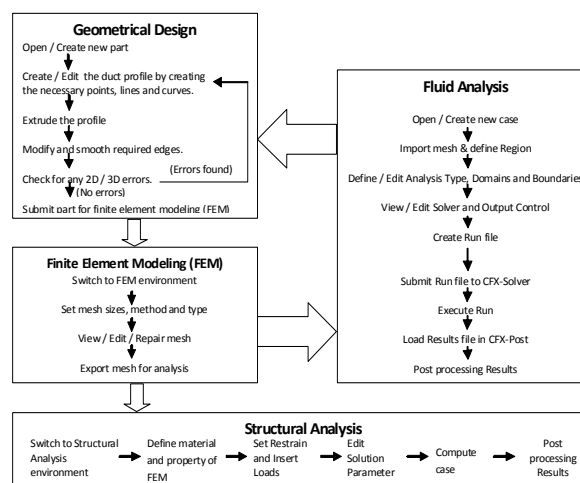


Fig. 1 A work flow applied in duct model

Once the meshing definitions are done, it can be executed and the completely built mesh presented in Fig. 2. The duct mesh created 18,755 nodes and 54,419 elements. The mesh product is then exported for analyses. From here onwards, the static (structural) and dynamic study is done with the help of simulation software, CATIA V5 for the static investigation

S. H. Yahaya is with the Faculty of Manufacturing Engineering, University of Technical Malaysia Melaka, 76100, Melaka, Malaysia (phone: 606-331-6494; fax: 606-331-6431; e-mail: saifudin@utem.edu.my).

J. M. Ali is with School of Mathematical Sciences, University of Science Malaysia, 11800, Penang, Malaysia (e-mail: jamaluma@usm.my).

M. R. Arham is with the Faculty of Manufacturing Engineering, University of Technical Malaysia Melaka, 76100, Melaka, Malaysia (e-mail: hafuza_hulawi@yahoo.com).

and ANSYS/CFX for the dynamic examination. The final work after analyzing is post processing the results. This is where tables and charts are plotted, visualization of Three-Dimensional streams and animation flows or reaction process can be made depending on the simulation software and the required user output.

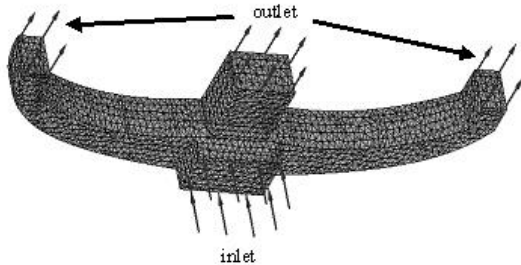


Fig. 2 Meshing process and airflow directions in duct model

III. STRUCTURAL ANALYSIS

A. Material Properties, Boundary and Loading Conditions

Base material of the duct is Polypropylene (PP), one of the lightest plastic materials, has a very good chemical resistance and an excellence fatigue resistance [1]–[2]. It is produced commercially in many different forms, depending on the properties desired. Mechanical properties of PP are strongly dependent on its crystallinity. Increasing crystallinity increases stiffness, yield stress and flexural strength but decreases toughness and impact strength [8].

This study initially wanted to employ PP with carbon fiber fillings. However, since interior automotive applications generally uses a talc-filled or mineral reinforced grade of PP and in choosing a material cost tops the concern list and is usually the overriding factor in deciding one material over the other [7]–[8]. We settled with a material from the Thermoplastic Olefins (TPO) family as this material is increasingly being used in automobile industries, replacing traditional engineering thermoplastics due to their low cost, performance properties and formulation flexibility [8]. Table I briefly describes the material properties used in this paper.

TABLE I
TPO CHARACTERISTICS

Young's Modulus (MPa)	Yield Strength (MPa)	Poisson Ratio
2480	17.5	0.36

In this case, boundary condition (BC) is set to Isostatic restraint to allow the modal deformation as a whole. Loading condition (LC) is defined to be forces distributed on the surfaces opposite the outlet as illustrated in Fig. 3. By varying the forces acting on the model, for example 500 N, the corresponding Von Mises Stress values can be obtained. Hence, safety factor (σ) is

$$\sigma = \frac{\text{Yield Strength of TPO}}{\text{Maximum Von Mises Stress}} \quad (1)$$

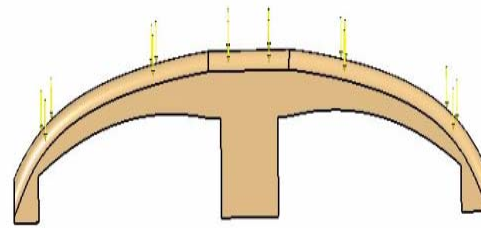


Fig. 3 Force directions placed along the duct surface

B. Result and Discussions

The static analysis is conducted and simulated after the material properties, BC and LC have been completely defined. The stress result is visualized in Fig. 4 as the LC applied is 500 N.

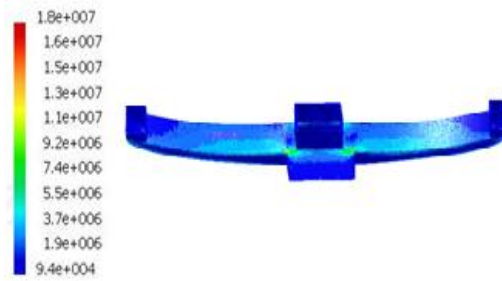


Fig. 4 Stress distributions occurred in duct model

Fig. 4 shows the Von Mises Stress distributions happened in duct model for the load equivalents to 500 N. It can be seen clearly that the maximum stress equals to 1.8E07 Pa (red color) while the minimum is 9.4E04 Pa (blue color). In engineering field, the design and material applied are considered safe to be used if $\sigma > 1$ while the failure condition occurred if $\sigma \leq 1$. Hence, the σ (load = 500 N) computed is 0.9722 by using (1). In this analysis, we also apply the various loads to find out when the design and material start to fail to be used as scheduled in Table II.

TABLE II
VARIOUS LOADS APPLIED AND ITS SAFETY FACTORS

Load (N)	Max. Von Mises Stress (Pa)	Safety Factor, σ
100	3.650E06	4.7945
200	7.300E06	2.3973
300	1.095E07	1.5982
400	1.460E07	1.1986
500	1.800E07	0.9722

In Table II, we can see that the design and material, TPO are in failure condition if load ≥ 500 N. However, we cannot simply say that the failure starts from this load. In order to know that, we apply an interpolation technique to estimate the

fatigue condition for this design and the material applied.

C. Fatigue Analysis

In engineering field, fatigue can be described as a failure condition happened on the structure design affected from the implications of repeated or varying loads. The intention of this section is to find out when the fatigue starts to occur in the design as shown in Fig. 2.

As mentioned earlier, the design fails to be used when the $\sigma \leq 1$. From this inequality, we understand that $\sigma < 1$ and $\sigma = 1$. Thus, we assume that the fatigue starts to happen when σ equals to 1. Next, will be the calculation on the related load due to this σ . In order to do that, the interpolation technique is applied. This technique is based on the Taylor series expansion and very useful in data prediction. Consider the following n th-order polynomial interpolation expressed as,

$$f_n(x) = a_0 + a_1(x - x_0) + a_2(x - x_0)(x - x_1) + a_3(x - x_0)(x - x_1)(x - x_2) + \dots + a_n(x - x_0) \dots (x - x_{n-1}) \quad (2)$$

where, n is the order of polynomial, a_0, a_1, \dots, a_n are the polynomial coefficients and the $(n + 1)$ data points, $\{x_i, f(x_i)\}$ with $i = 0, 1, 2, 3, \dots, n$. These coefficients can be evaluated using all these points. Equation (2) also known as the n th-order Newton's divided-difference interpolating polynomial. Let develop the first-order Newton interpolation ($n = 1$). Thus, we need two data points and obtain,

$$a_0 = f(x_0), a_1 = \frac{f(x_1) - f(x_0)}{x_1 - x_0} \quad (3)$$

Then, the first-order Newton's polynomial can be written as,

$$f_1(x) = a_0 + a_1(x - x_0) \quad (4)$$

By referring to Table II, we set the total loads as in x -axis and safety factor as in y -axis or $f(x)$ because x -axis is known as an input factor while y -axis as an output factor. As explained before, the design and material applied fail to be used between the range of 400 N to 500 N. Thus, a_0 and a_1 can be computed by using (3) where,

$$a_0 = 1.1986, a_1 = -2.2640E - 03 \quad (5)$$

These values in (5) can be used in developing the first-order Newton's polynomial as,

$$f_1(x) = 1.1986 - 2.2640E - 03 * (x - 400) \quad (6)$$

By setting $f_1(x) = 1$, we solve (6) to estimate,

$$x \approx 487.7208 \text{ N} \quad (7)$$

According to this analysis, the design and material applied are considering in safety mode if the load < 487.7208 N while the fatigue of this duct design starts to happen if the load ≥ 487.7208 N.

IV. FLUID ANALYSIS

CFD codes are structured around numerical algorithms that can solve the fluid flow problems [9]. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion. Such a mesh can be either irregular or regular. Some of the discretization methods being used such as Finite volume method (FVM), Finite element method (FEM) and Finite difference method (FDM) [6], [10].

Most of the car makers today invent an instrument panel with three air outlets; at the center, left and right side of the panel. The design proposes in this paper is based on the standard structure. Upon completion of the geometrical (CAD) and finite element model (mesh) of the duct, it is then exported to CFX-Pre where all the simulation parameters are defined. Fundamental settings include steady state analysis type, fluid domain with air as the fluidic matter and non-turbulent (laminar) flow. Wall Boundary Condition (BC) is defined as no slip wall and heat transfer is set to Adiabatic. For inlet BC, the velocity is set to 3 m/s with a temperature of 25 °C. Meanwhile for the remaining outlet BC, average static pressure is chosen with relative pressure valued as nil and pressure averaging is set to over whole outlet.

FEM imported in CFX-Pre along with the direction of inlet and outlets (Fig. 2). The property of air is not defined in this CFX because it is already characterized internally under CFX Library Materials. As a result, see in Fig. 5.

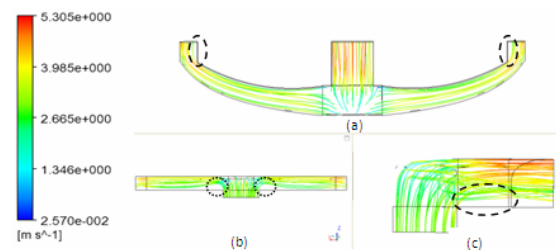


Fig. 5 Velocity of airflow in duct model with different views

Fig. 5 illustrates the velocity of airflow from the duct inlet through its outlet. Notice that the air velocity at the outlet is higher than the air velocity at the inlet of the duct. This is due to the nonzero acceleration of the fluid particle as described in (8) such,

$$\tilde{a} \equiv \frac{DV}{Dt} = (V \cdot \Delta)V + \frac{\partial V}{\partial t} \quad (8)$$

with

$$\left\{ \frac{DV}{Dt}, (V \cdot \Delta)V, \frac{\partial V}{\partial t} \right\} \quad (9)$$

Equation (9) has the notations of acceleration named as total particle, convection and local. Since velocity field is a function of time, then a fluid particle is in unsteady state where local acceleration is nonzero acceleration while the steady state fluid flow, the local becomes zero and only convection acceleration exist due to fluid particle motion [3], [5]. The dotted elliptical in Fig. 5 explains a stagnation area or the recirculation flow occurrence which is caused by sharp corners. Flow separation happened at these corners and hence, the vena contracta region is formed. The fluid velocity increases at this region because of the reduced flow area and decreases as the flow fills the entire area [3], [5].

Next finding is the pressure contour generated all over the duct surface as shown in Fig. 6. It can be seen clearly that the center of the duct has the highest pressure load compared to any other parts of the duct. This is due to the inlet flow. However, it is an insignificance value compared to the yield strength of TPO with the maximum pressure highly accurate to 15 Pa.

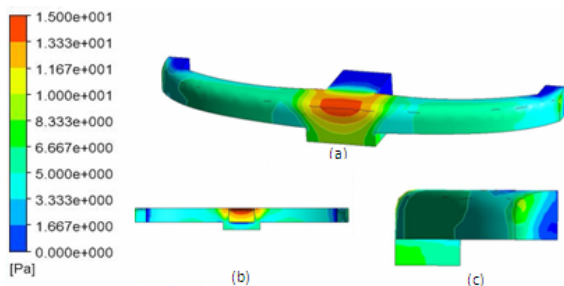


Fig. 6 Pressure Contour in duct model with different views

V. CONCLUSION AND RECOMMENDATIONS

In this paper, the duct design with TPO has done all the analyses such as static, fatigue and dynamic analysis. As the results, we can clearly seen that the design with this TPO is in safety condition if the load < 487.7208 N. In dynamic approach, ANSYS CFX is chosen as a tool for the CFD simulation respectively. It shows that the duct should be eliminated the sharp edges at the inlet, bends or branches by creating a more rounded edge as to avoid the recurring of vena contracta regions. Normally, the safety factor of duct model, $\sigma \geq 10$; nevertheless, another study should be conducted to determine the impact capabilities and energy absorption potential of the design and material chosen. The current simulation environment is performed by using steady-state conditions. In future, we would like to include a transient simulation for thermal analysis, to model the fluid as a

turbulent flow and to apply the application of Computer Aided Geometrical Design (CAGD) especially the circle to circle templates in the design.

ACKNOWLEDGMENT

This research is supported by the Department of Higher Education, Malaysia and UTeM under FRGS grant. The authors gratefully acknowledge to everybody for their helpful comments.

REFERENCES

- [1] K. G. Budinski and M. K. Budinski, *Engineering Material: Properties and Selection*, 9th Edition, Pearson, 2010.
- [2] D. A. Chasis, *Plastic Piping System*, 2nd Edition, Industrial Press Inc, 1988.
- [3] J. M. Cimbala and Y. A. Cengel, *Essentials of Fluid Mechanics: Fundamentals and Applications*, McGraw Hill, 2008.
- [4] P. Dvorak, "Machine Design Magazine," *Penton Media Inc*, vol. 36, 2006.
- [5] R. W. Fox, *Introduction to Fluid Mechanics*, 6th Edition, John Wiley & Sons Inc, 2004.
- [6] T. Glatzel, C. Litterst, C. Cupelli, T. Lindermann, C. Moosmann, R. Niekrawietz, W. Streule, R. Zingerle and P. Koltay, "Computational fluid dynamics (CFD) software tools for microfluidic applications – A case study," *Computer & Fluids*, vol. 37, 2008, pp. 218-235.
- [7] J. A. Jacobs and T. F. Kilduff, *Engineering Material Technology: Structures, Processing, Properties and Selection*, 5th Edition, Prentice Hall, 2005.
- [8] C. Maier and T. Calafut, *Polypropylene: The Definitive User's Guide and Databook*, Plastic Design Library, 1998.
- [9] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*, 2nd Edition, Pearson, 2007.
- [10] S. Wannarumon and M. A. Grande, "Comparisons of Computer Fluid Dynamic Software Programs Applied to Jewelry Investment Casting Process," *World Academy of Science, Engineering and Technology*, vol. 55, 2009, pp. 88-95.