

## Understanding of finite element analysis to analyze the air flow in an air purifier

Ritwik Tiwari<sup>1</sup>, Niyati Raut<sup>2</sup>

<sup>1</sup> Department of Mechanical Engineering, VIVA Institute of Technology, Virar, Maharashtra, India

<sup>2</sup> HOD, Department of Mechanical Engineering, VIVA Institute of Technology, Virar, Maharashtra, India

### Abstract

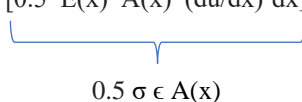
The novel finite element analysis (FEA) method simulates different parts or the entire assembly to test the system under real-world operating circumstances. This technique is utilized by all manufacturers and designers across a wide range of industries, including aircraft, automotive, steel, drones, etc., to minimize errors or defects that arise during the production or designing stage. Engineers can forecast how structures and component behavior will change under various loading scenarios with the use of FEA. When it comes to creating and optimizing things before they are actually constructed, this predictive power is essential. To make sure that the finished product satisfies performance and safety requirements, engineers might examine elements like stress, strain, and deformation. In order to improve the air quality output from the purifier, the study uses FEA to analyze the air flow through the air purifier's casing by establishing the working boundary condition and measuring the deflection along various directions. The procedure gave us a better air purifier output by allowing us to adjust the air purifier's casing thickness appropriately to obtain higher-quality air without significantly altering the device's overall design.

**Keywords:** Finite element analysis (FEA), mesh generation, boundary conditions, displacement analysis, fatigue analysis, mesh quality

### Introduction

In the early 1960s, Ray R. Clough invented Finite Element Analysis (FEA), a potent engineering technique that helps us understand how structures behave under various conditions. His book, "The Finite Element method in plane stress analysis," revolutionized the manufacturing industry by helping us understand how materials behave under various conditions. Finite elements are the smaller, more manageable components that are used to dissect complex objects. Every component is examined separately, and their interconnections shed light on the behavior of the complete structure. Engineers are able to forecast the behavior of structures in different environments and stresses by applying mathematical formulas and simulations. In order to ensure that everything is safe and dependable under various conditions, from buildings and bridges to airplanes and automobile parts, finite element analysis (FEA) is essential.  $u(x) = u_h(x) + e(x)$  This is subsequently investigated once again by scientists such as Schellbach (1851) and Courant (1943), after which Turner, Clough, Martin, Topp (1956), Argyris (1957), Babushka & Aziz (1972) evaluate the data in real-world situations. In order to construct the structure required for the examinations, FEA works by enabling the simulation to transform an object into a mesh of millions of elements. Each element is calculated separately, and there are a number of locations known as Nodal points that are crucial for meshing using formulae like

$$A_i = \int [0.5 * E(x) * A(x) * (du/dx)^2 dx]$$


  
 $0.5 \sigma \epsilon A(x)$

Then external force equation like

$$A_a = A(x) * t(x) * u(x) | \Gamma_t$$

Without compromising the object's structural design, mesh convergence is one of the most difficult tasks in the process. As Fig 1 illustrates, this has become a crucial task since it requires us to comprehend different degrees of freedom in order to counter converge in quality.

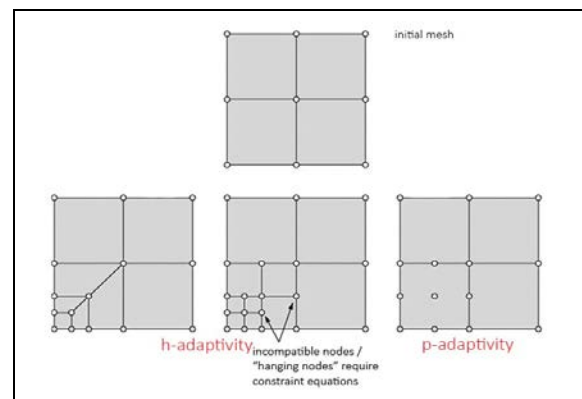


Fig 1: Mesh Refinement



Fig 2: Design dimension of Air Purifier

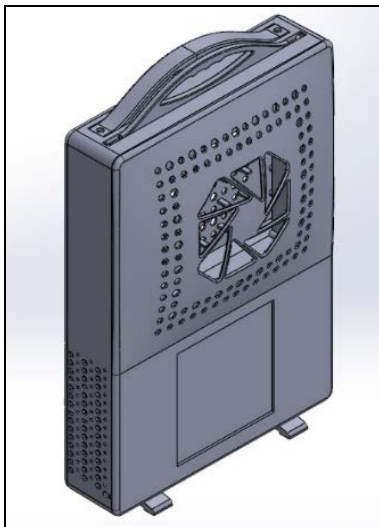


Fig 3: CAD model of Purifier

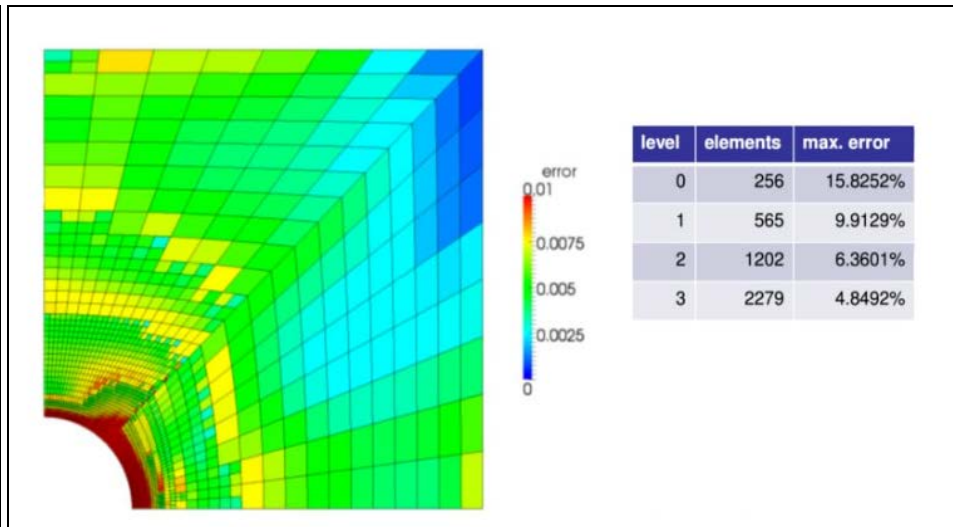


Fig 4: Application of Mesh Refinement

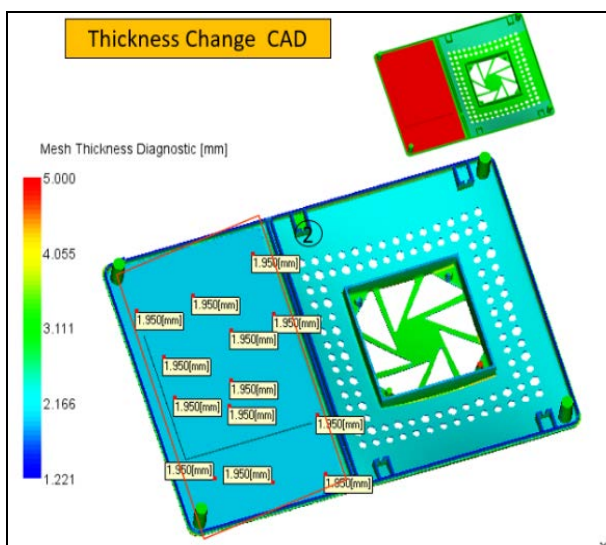


Fig 5: Mesh Generation

Optical Properties	Environmental Impact	Material data completeness	Cr
Description	Recommended Processing	Rheological Properties	Thermal Prop
Family name	ACRYLONITRILE COPOLYMERS (ABS, ASA, ...)		
Trade name	Absolac 100		
Manufacturer	INEOS Styrolution		
Link			
Family abbreviation	ABS		
Material structure	Amorphous		
Data source	Moldflow Corporation : pvT-Measured : mech-Supplementa		
Date last modified	08-AUG-13		
Date tested	01-NOV-99		
Data status	Non-Confidential		
Material ID	54402		
Grade code	BA001		
Supplier code	STYROLUT		
Fibers/fillers	Unfilled		

Fig 6: Testing Material Details

Convergence is now investigated using a variety of formulas provided by Jacob Fish & Ted Belytschko (2007), R Courant (1943), and K Schellbach such as Displacement Error:

$$e_u = u - u_h$$

where  $u$  is the analytical solution for the displacement field.

Strain Error:

$$e_\epsilon = \epsilon - \epsilon_h$$

where  $\epsilon$  is the analytic solution for the strain field.

Stress Error:

$$e_\sigma = \sigma - \sigma_h$$

where  $\sigma$  is the analytic solution for the stress field.

Subsequently, the object undergoes a detailed analysis to improve its quality and output for the system by applying different working and boundary conditions.

A number of technologies are available to improve our understanding of structures and their behaviors in addition to Finite Element Analysis (FEA). Adams and Sim pack are two examples of multibody dynamics (MBD) software that makes it easier to analyze large mechanical systems by simulating the motion and interactions of connected rigid or flexible bodies explained thoroughly by Kevin Schmichen, Florian Schwaiger, Maximilian A. Wechner and Florian Holzapfel(2022), however Without having to spend the time

and money necessary to build a prototype and conduct physical testing, finite element methods allow us to swiftly iterate through a large number of alternative design configurations in order to uncover possible issue areas or ways we may improve the design. Beyond finite element analysis (FEA), scientist like P Ladevèze (2012) also highlights the parallel and non-parallel approach for non-incremental approach for the calculation. computational structural mechanics (CSM) software, such as Abaqus or LS-DYNA, covers a wider variety of structural analyses, such as explicit dynamics and material failure simulations. Engineers may model and analyze structures using a variety of techniques, such as matrix analysis, finite element analysis, and finite difference analysis, thanks to structural analysis and design software, which is exemplified by applications like SAP2000 or STAAD.Pro but flexibility of Finite Element Analysis (FEA) in managing intricate structures and a wide range of material properties gives it an advantage over Classical Structural Mechanics (CSM). FEA is skilled at interpreting complex geometry and taking into account a range of loading scenarios. In particular, when handling nonlinear material behaviors, it offers a more realistic portrayal of real-world situations. As a result, FEA has emerged as the gold standard for analyzing the air

purifier's casing thickness in order to improve airflow while requiring the least amount of time and money to operate and produce higher-quality results.

## Materials & Methodology

Step 1: Problem Definition: Define analysis objectives and scope.

Step 2: Geometry Creation: Develop a 3D CAD model of the air purifier.

Step 3: Mesh Generation: Divide the model into finite elements for analysis.

Step 4: Material Properties: Assign mechanical and thermal properties to components.

Step 5: Boundary Conditions: Specify constraints, loads, and operating conditions. Analysis Type: Select the appropriate FEA analysis (static, dynamic, thermal, FSI). Solver Setup: ConFig solver settings for convergence and accuracy.

Step 6: Analysis Execution: Run FEA simulations to obtain results. Results Post-Processing: Analyze stress, deformation, temperature, airflow, etc. Optimization Iterations: Make design changes based on results for performance optimization.

Step 7: Validation and Verification: Compare FEA results with physical testing data. Documentation: Document methodology, assumptions, and results. Iterative Improvement: Continuously refine design and FEA models as needed to meet performance and safety goals.

Finite Element Analysis (FEA) is a numerical simulation method, not an experimental procedure. However, you can conduct experimental testing to validate FEA results or gather data needed for an FEA analysis.

### Step 1: Problem Definition

The primary objective of utilizing Finite Element Analysis (FEA) for heat conduction analysis in an air purifier is to comprehensively assess the thermal performance of the device. By simulating the heat conduction within various components of the air purifier, including filters, fans, and electronic elements (shown in Fig 2), FEA aids in understanding how these parts behave under diverse operational conditions. This step aims to pinpoint potential hotspots within the device where excessive temperatures might occur, posing risks to both the air purifier's functionality and longevity. FEA also plays a crucial role in optimizing heat dissipation strategies, allowing engineers to design effective thermal management solutions that prevent critical components from exceeding safe temperature limits. Moreover, by ensuring that the air purifier maintains thermal reliability, FEA contributes to energy efficiency goals by minimizing cooling-related energy consumption.

### Step2: Geometry Creation

Geometry creation is a fundamental step in Finite Element Analysis (FEA), where a digital representation of the physical structure or component under analysis is generated. This process often begins with a Computer-Aided Design (CAD) model (shown in Fig 3), which provides an accurate starting point. CAD software allows engineers to create precise 3D representations, capturing intricate details of the object. However, if a CAD model is unavailable, engineers may need to create a simplified geometric representation directly within the FEA software.

### Step 3: Mesh Generation: Divide the model into finite elements for analysis.

Mesh generation is a critical step in Finite Element Analysis (FEA) where the digital model of the physical structure or component is divided into finite elements to facilitate analysis. This process involves discretizing the geometry, effectively breaking it down into smaller, manageable elements that approximate the shape and behavior of the real-world object. As seen in Fig 1 above, there are two different kinds of refinements: h-refinement and p-refinement. P-refinement refers to raising the element's order, whilst H-refinement deals with decreasing the element sizes. Particularly when meshing a curved surface, it is crucial to discern between geometric effect and mesh convergence in this situation. To precisely capture the boundary, more elements (or mesh refinement) will be needed when using straight (or linear) elements (shown in Fig 4 & 5).

Errors are significantly reduced by mesh refinement. In any analysis, the equilibrium equations are fundamental. They state that the sum of forces and moments acting on a structure in equilibrium must be equal to zero. These equations form the basis for most types of FEAS.

Forces Equilibrium (in 2D):

$$\sum F_x = 0$$

$$\sum F_y = 0$$

Moments Equilibrium (in 2D):

$$\sum M = 0$$

isotropic materials, Hooke's Law is commonly used:

$$\sigma = E \cdot \epsilon$$

Where:

$\sigma$  is the stress in the material (force per unit area),

$E$  is the Young's Modulus of the material (a material property), and  $\epsilon$  is the strain in the material (dimensionless deformation). The choice of element type (e.g., tetrahedral, hexahedral, or other types) and mesh size is essential. Engineers must strike a balance between accuracy and computational efficiency. Finer meshes offer more precise results but demand greater computational resources.

Let's consider an example of a simple 1D truss element to demonstrate the formulation of element equations in Finite Element Analysis (FEA).

- Young's Modulus ( $E$ ) of the material:  $2 \times 10^5 \text{ N/mm}^2$
- Cross-sectional area ( $A$ ) of the truss member:  $100 \times 2100 \text{ mm}^2$
- Length ( $L$ ) of the truss member:  $1000 \text{ mm}$
- Applied force ( $F$ ) at node  $i$ :  $5000 \text{ N}$
- Displacement at node  $j$ :  $u_j = 5 \text{ mm}$

Steps for Formulating Element Equations:

#### 1. Interpolation

Let  $u_i$  and  $u_j$  represent the nodal displacements at nodes  $i$  and  $j$  respectively, interpolation function for a linear element in 1D is  $N_1(\xi) = 0.5(1 - \xi)$  and  $N_2(\xi) = 0.5(1 + \xi)$ , where  $\xi$  varies from  $-1$  to  $+1$  along the element.

#### 2. Strain-Displacement Relations

The axial strain ( $\xi$ ) is given by:

$$\xi = \frac{du}{d\xi} = \frac{u_i - u_j}{L} = \frac{5 \text{ mm} - 0}{1000} = 0.005$$

### 3. Constitutive Equation

Using Hooke's law, Stress ( $\sigma$ ) can be calculated by:  
 $\sigma = E \cdot \epsilon = (2 \cdot 10^5) \text{ N/mm}^2 \cdot 0.005 = 1000 \text{ N/mm}^2$

### 4. Equilibrium Equation

The axial force (F) in the truss member is given by:  
 $F = A \cdot \sigma = 100 \text{ mm}^2 \cdot 1000 \text{ N/mm}^2 = 100000 \text{ N}$

### 5. Assembly

The stiffness matrix (K) for a 1D truss element with two nodes and one DOF per Node is given by:

$$K = \frac{EA}{L} \begin{bmatrix} 1 & 1 \\ -1 & 1 \end{bmatrix}$$

Substituting the given values ( $E = 2 \cdot 10^5 \text{ N/mm}^2$ ,  $A = 100 \text{ mm}^2$ ,  $L = 1000 \text{ mm}$ ),  
 We get:

$$K = \begin{bmatrix} 2000000 & -2000000 \\ -2000000 & 2000000 \end{bmatrix} \text{ N/mm}$$

### Step 4: Material Properties: Assign mechanical and thermal properties to components.

Assigning material properties to components is a pivotal step in Finite Element Analysis (FEA), as it involves specifying the mechanical and thermal characteristics of the materials utilized within the analysis (shown in Fig 6). In FEA, materials are typically described by their mechanical properties, which encompass parameters such as Young's Modulus, Poisson's Ratio, Yield Strength, and Ultimate Strength. Young's Modulus characterizes a material's stiffness or elasticity, dictating how much it will deform when subjected to external forces. Poisson's Ratio informs how a material's cross-sectional dimensions change when subjected to axial deformation, offering insights into its deformation behavior. Yield Strength and Ultimate Strength describe the material's ability to withstand mechanical loading before undergoing plastic deformation or failing. Additionally, in thermal FEA, material properties extend to thermal conductivity, specific heat, and thermal expansion coefficients, which govern how materials conduct and store heat.

### Step 5: Boundary Conditions: Specify constraints, loads, and operating conditions. Analysis Type: Select the appropriate FEA analysis (static, dynamic, thermal, FSI). Solver Setup: ConFig solver settings for convergence and accuracy.

In Finite Element Analysis (FEA), boundary conditions, analysis type, and solver setup are pivotal elements that collectively shape the accuracy and effectiveness of the simulation. Boundary conditions encompass the constraints, loads, and operating conditions that dictate how the structure or component interacts with its environment. Constraints specify fixed or immovable points, edges, or surfaces, effectively anchoring the model, while loads introduce external forces, pressures, or thermal conditions that mimic real-world operating scenarios as shown in Fig 7. Properly defining these boundary conditions is essential for ensuring that the FEA accurately simulates the desired conditions and responses.

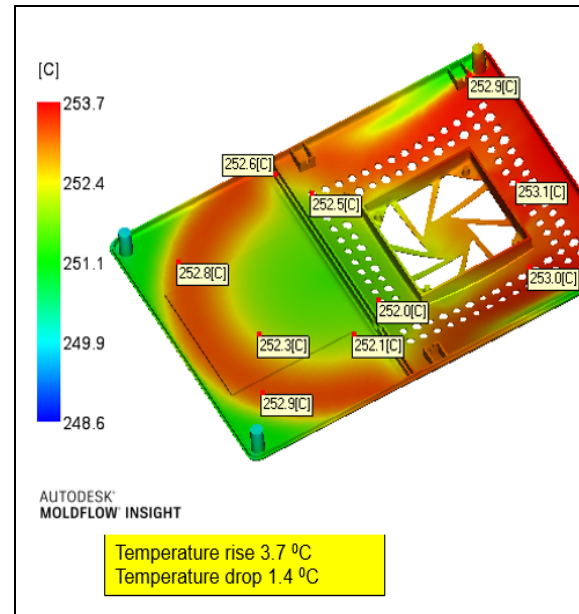


Fig 7: Boundary definition

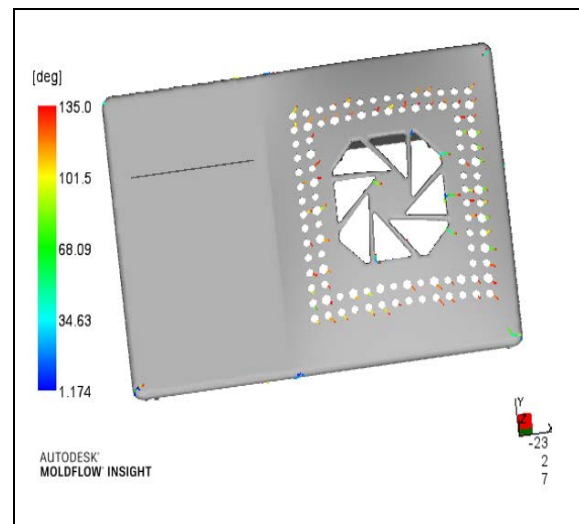


Fig 8: Post Processing Airflow Analysis

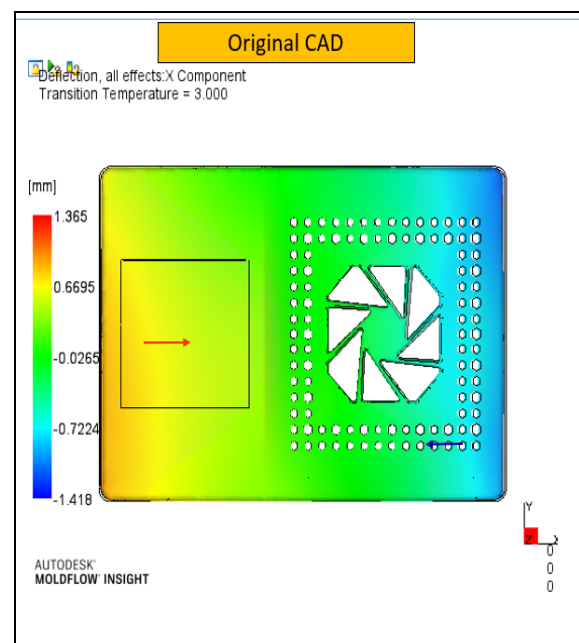


Fig 9: Validation and Verification of Airflow Analysis part 1

**Step 6: Analysis Execution: Run FEA simulations to obtain results. Results Post-Processing: Analyze stress, deformation, temperature, airflow, etc. Optimization Iterations: Make design changes based on results for performance optimization.**

Following the setup of the Finite Element Analysis (FEA) model, the analysis execution phase comes into play, serving as the pivotal moment when the model is subjected to the specified conditions, and the FEA software employs numerical algorithms to solve the governing equations. This process generates a wealth of data pertaining to critical factors such as stress distribution, deformation, temperature profiles, or airflow patterns, depending on the focus of the analysis. Once the simulation is complete, the subsequent results post-processing stage involves scrutinizing and interpreting the extensive dataset. This step is essential for extracting meaningful insights into how the structure or system behaves under various conditions. Engineers' analyses stress levels to identify potential failure points, evaluate deformations to ensure design compliance, assess temperature distribution for thermal management, and examine airflow patterns for effective ventilation as shown in Fig 8.

After scrutinizing the results, the iterative optimization process begins. Optimization iterations involve making informed design changes based on the obtained data, aiming to enhance overall performance, efficiency, and safety. Engineers can adjust material properties, alter geometric configurations, or modify boundary conditions to optimize the design. These iterations may involve a series of refinements and simulations until the desired performance goals are met, all while ensuring that the design remains within safety margins. Collectively, these steps in the FEA process—analysis execution, results post-processing, and optimization iterations—enable engineers to make informed design decisions, refine their designs, and ultimately achieve the desired performance, reliability, and efficiency in their engineering projects.

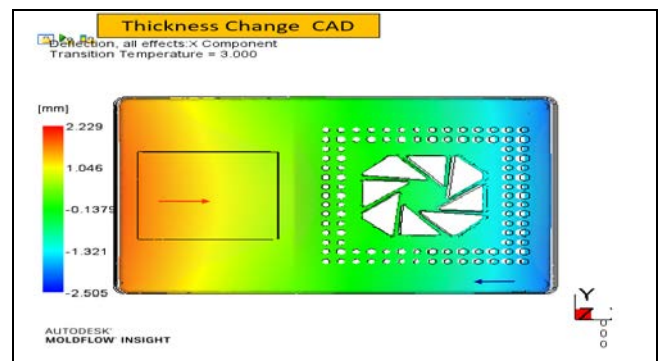
**Step 7: Validation and Verification: Compare FEA results with physical testing data. Documentation: Document methodology, assumptions, and results. Iterative Improvement**

Continuously refine design and FEA models as needed to meet performance and safety goals.

Following the completion of Finite Element Analysis (FEA) simulations and the documentation of results, the process moves to validation and verification. During this phase, engineers rigorously compare the FEA results with data gathered from physical testing or real-world observations. By conducting this comparison, engineers can assess the accuracy and reliability of the FEA simulations. Validating the FEA model involves ensuring that it accurately replicates the physical behavior of the system or component under various conditions, while verification confirms that the FEA software and model are functioning correctly. This step is crucial in building confidence in the FEA results and ensuring that the virtual representations align with real-world behavior as shown in Fig 9 & 10.

Simultaneously, thorough documentation remains a central pillar of the FEA process. Engineers meticulously record the methodology, assumptions, and results of the analysis, providing a clear and transparent account of the entire process. Documentation serves multiple purposes, including

facilitating collaboration among team members, ensuring repeatability of the analysis, and providing a basis for future reference and analysis improvements.



**Fig 10:** Validation and Verification of Airflow Analysis part 2

**Result**

Air flow is anticipated in this work by using the numerical finite element analysis (FEA) that was created. By comparing the numbers produced by the numerical simulation with those acquired through experimentation or analytical computation, the accuracy of the generated values is verified. The comparative study of the air flow and the experimental findings of temperature distribution and the impact of material and manufacturing process variation on air flow are developed in the following sections. The purifier's enclosure was thicker for model manufacture, as was previously reported. Based on this supposition, we proposed that the casing's cooling and air flow were not evenly distributed. Because of this, the temperature predictions on the casing are overestimated in the regions where the bodies overlap. Additionally, a significant volume of material with inadequate air movement was produced, which resulted in the heating of the purifier's body. We were able to investigate the purifier's casing accurately through the application of FEA, ensuring that the temperature of the purifier body remains constant and that the air flow distribution is appropriate. The design modifications are detailed below.

**Table 1:** Consolidated Results

Parameter	Option 1 (original Cad)	Option 2 (Thickness changed Cad)
Fill time (sec)	3.333	2.450
Total pressure (MPa)	56.35	54.23
Cavity Pressure (MPa)	39.89	37.34
Temperature at flow Front (°C)	4.5	4.2
Tonnage (T)	394	350

**Conclusion**

Applications of Finite Element Analysis (FEA) in science and engineering provide a plethora of insightful findings. First of all, FEA helps to ensure the functioning and dependability of structures and systems by offering a thorough understanding of how they operate under various circumstances. It is an effective optimization technique that lets engineers refine designs through iterative testing of various configurations, materials, and loads. In addition to improving performance, this optimization procedure lowers costs. Furthermore, FEA plays a critical role in maintaining safety by helping to identify possible failure modes, supporting the design of structures with strong safety

margins, and producing a thorough analysis of air flow through the air purifier's casing for improved air quality.

### Acknowledgement

Authors are profoundly thankful to the editors and the anonymous reviewer for their productive comments which led to improvements in the article.

### References

1. An Introduction to Nonlinear Finite Element Analysis Second Edition: with applications to heat transfer, fluid mechanics, and solid mechanics JN Reddy, 2014.
2. Kevin Schmiechen, Florian Schwaiger, Maximilian A Wechner, Florian Holzapfel. "Combining ALM and MBD Tools for Continuous Requirement Validation Tests with Multi-Dimensional Test Parameters", 2022.
3. COCK RD. "Concepts and Applications of Finite Element Analysis", forth edition wiley india limited 2018, 1-430.
4. Yingjun Wang1, Zhenpei Wang, Zhaohui Xia, Leong Hien Poh. "Structural Design Optimization Using Isogeometric Analysis: A Comprehensive Review", Tech Science Press CMES, 2018:117:455-507.
5. TJR Hughes, JA Cottrell, Y Bazilevs. "Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement", Computer Methods in Applied Mechanics and Engineering, 2005:194(39-41):4135-4195.
6. Masakazu T. "The finite element analysis about size effect of concrete members", Memoirs of the Faculty of Engineering, Fukuyama University, 2001:25:49-54.
7. Jacob Fish, Kamlun Shek, Muralidharan Pandheeradi, Mark S Shephard. "Computational plasticity for composite structures based on mathematical homogenization: Computer Methods in Applied Mechanics and Engineering, 1997:148(1-2):53-73.
8. TA Laursen. "Computational contact and impact mechanics: fundamentals of modeling interfacial phenomena in nonlinear finite element analysis", ISBN 3-540-42906-9 springer-verlag berlin Heidelberg newyork 2003.
9. Roth S, Oudry J. "Influence of mesh density on a finite element model under dynamic loading", Proceedings of 3 rd European Hyperworks Technology Conference, November 2nd-4th, 2009, Ludwigsburg, Germany.
10. Li Z P, Kindig M W, Subit D, Kent R W. "Influence of mesh density, cortical thickness and material properties on human rib fracture prediction", Medical Engineering & Physics, 2010:32(9):998-1008.
11. Liu YC, Day ML. Simplified modeling of thinwalled box section beam", International Journal of Crashworthiness, 2006:11(3):263-272.
12. Ashby MF, Evans A, Fleck NA. Metal Foams: A Design Guide, Butterworth Heinemann, 2000.
13. Huiskes R, Chao EYS. A survey of finite element analysis in orthopedic biomechanics: the first decade, Journal of Biomechanics, 1983:16: 385-409.
14. Verma RJ. Melosh, Numerical tests for assessing finite element model convergence, International Journal for Numerical Methods in Engineering, 1987:24:843-857.
15. Cifuentes AO, Kalbag A. A performance study of tetrahedral and hexahedral elements in 3-D finite element structural analysis, Finite Elements in Analysis and Design, 1992:12:313-318.
16. Benzley SE, Perry E, Merkley K, Clark B, Sjaardema G. A Comparison of AllHexahedral and All-Tetrahedral Finite Element Meshes for Elastic and ElastoPlastic Analysis, Proceedings, 4th International Meshing Roundtable, Sandia National Laboratories, 1995, 179-191.
17. Viceconti M, Bellingeri L, Cristofolini L, Toni A. A comparative study on different methods of automatic mesh generation of human femurs, Medical Engineering & Physics, 1998:20:1-10.
18. Hoskins MH, Kunz RF, Bistline JE, Dong C. Coupled flow-structurebiochemistry simulations of dynamic systems of blood cells using an adaptive surface tracking method, Journal of Fluids and Structures In Press, Corrected Proof, 2009.
19. MSC.Marc, User Documentation Marcs 2007 r1 Volume B: Element Library, in, 2007.
20. VDMA Photovoltaic Equipment, International Technology Roadmap for Photovoltaic (ITRPV): Results 2018 including maturity report 2019, 10th edition, 2019.
21. Lorenz M, Linse H, Frintrup M, Jeitler A, Mette M, Lehner R, Greutmann H. *et al.* Clement, 35th European Photovoltaic Solar Energy Conference and Exhibition, 2018, 819.
22. Thibert S, Jourdan J, Bechevet B, Faissat J, Mialon S, Chaussy D, *et al.* "Combining design of experiments and power loss computations to study the screen printing process," in 2013 IEEE 39th Photovoltaic Specialists Conference (PVSC): 16 - 21 June 2013, Tampa, Florida (IEEE, Piscataway, NJ, 2013), p. 2280.
23. Cabos C, Matthies HG. The energy finite element method noise FEM, in: A.K. Belyaev, R.S. Langley (Eds.), Proc. IUTAM Symp. Vib. Analysis of Structures with Uncertainties, St Petersburg, 2009, in: IUTAM series, vol. 27, Springer, Heidelberg, 2011.
24. Chappell DJ, Giani S, Tanner G. Boundary element dynamical energy analysis: a versatile method for solving two or three dimensional wave problems in the high frequency limit, J. Comput. Phys, 2012:231:6181-6191.