Development of Thermal and Structural Deformation Model to Predict the Part Build Dimensional Error in Fused Deposition Modeling

K. Chockalingam*
Department of Mechanical Engineering,
Thiagarajar College of Engineering, Madurai-15, Tamilnadu, India
*Email: kcmech@tce.edu
*Corresponding author

M. Karthic
Department of Mechanical Engineering,
Thiagarajar College of Engineering, Madurai-15, Tamilnadu, India

N. Jawahar
Department of Mechanical Engineering,
Thiagarajar College of Engineering, Madurai-15, Tamilnadu, India

Received: 29 January 2018, Revised: 6 February 2018, Accepted: 6 April 2018

Abstract: The most common extrusion based technology in rapid prototyping is Fused Deposition Modeling (FDM). In FDM process, widely used materials are Acrylonitrile Butadiene Styrene (ABS) and Polycarbonate. In this study ABS-P430 material is considered. During the part build process, the rapid heating and cooling is happening on the build part which leads to high thermal gradient. This thermal gradient causes thermal stress; it will lead to deformation of build parts. In this paper a three dimensional transient thermo-mechanical Finite Element Analysis (FEA) had been used to find out the maximum principal stress and deformation of the build part. This FEA analysis is called as thermal and structural deformation model or 3D FEA model. In this model, the novel technique called Element birth/death is used in ANSYS11 to mimic the FDM process. The most influencing parameters of FDM process called orientation and layer thickness have been considered in a 3D FEA model to calculate the deformation of a part. To validate the work, a standard design which is considered in 3D FEA model is fabricated using dimension 1200es FDM machine using same orientation and layer thickness and deformation is measured. From the results it was observed that the relative error between 3D FEA model and actual fabricated model is found to be 3-6%. This 3D FEA model would be helpful for RP machine users to find the deformation of the build part before making the products.

Keywords: Deformation, Deformation model, Fused deposition modeling, Thermal and structural element birth/death function

Reference: Chockalingam, K., Karthic, M. and Jawahar, N., "Development of Thermal and Structural Deformation Model to Predict the Part Build Dimensional Error in Fused Deposition Modeling", Int J of Advanced Design and Manufacturing Technology, Vol. 11/No. 2, 2018, pp. 27-37.

Biographical notes: K. Chockalingam is associate professor in the Department of Mechanical Engineering, Thiagarajar college of Engineering, Madurai. His research interests are process optimization, Rapid prototyping and Bio-Medical Engineering. **M. Karthic** is assistant professor in the department of mechanical engineering at Thiagarajar College of engineering, Madurai. His current research focuses on Additive Manufacturing. **N. Jawahar** is professor in the department of mechanical engineering and dean (R&D) in Thiagarajar College of Engineering. His academic and research interest are process optimization, production planning and control, evolutionary programming and supply chain management.

1 INTRODUCTION

Additive Manufacturing (AM) technology is widely used for fabricating parts layer by layer directly from a CAD data in the form of STL format. The process builds objects by adding material in a layer by layer manner to create a three-dimensional (3D) part, provides the benefit to produce complex parts with lower cost and shorter cycle time compared to other conventional manufacturing process [1] – [2]. There are many commercial additive manufacturing systems available in global market such as Fused Deposition Modeling (FDM), 3D printing, selective laser sintering (SLS), stereo-lithography (SLA), Direct Metal Deposition (DMD) and inkjet modelling (IJM). These systems differ in the manner of building layers, power sources and in the types of materials that can be fabricated by these processes safely [3].

In AM technology the most widely used process is Extrusion-based fabrication method. Currently the market for extrusion based AM technology is fused deposition modeling (FDM) process and machines from Stratasys, Inc [17]. In FDM process mostly used thermoplastics materials are Acrylonitrile Butadiene Styrene (ABS) and Polycarbonate (PC). In FDM process extruded material temperature is 300^0C and chamber temperature is 80^0C due to these rapid heating and cooling of material happened by conduction heat transfer and forced convection between material and chamber. This will lead to development of thermal stress during part build process in FDM process. These thermal stresses lead to increase in principal stress and deformation of the build part. This deformation of the part will cause inaccuracies in dimensions and affect the functionality of the build part.

Modern manufacturing industries such as telecommunication, electronics and aerospace require very minimum dimensional error [13]. The dimensions of the FDM fabricate part greatly depends on the deformation of the part. So the development of deformation modelling is essential to predict the dimensional error of the build part. The two possible approaches are available to predict the dimensional error of FDM process. Development of thermal and structural deformation model / 3D Finite Element Analysis (FEA) model to simulate the FDM process, so as to obtain the deformation data and optimization of process parameters during the fabrication stage so that to minimize the deformation of the product by selecting suitable process parameters. Selecting the optimal process parameters is consuming more time and money. Hence to develop the deformation modelling with the available options, 3D FEA model is the best possible solution to predict the dimensional error.

Several attempts have been made to develop the deformation modelling with adjustment of process parameters of FDM process and other AM technology by different researchers. Zhang and Chou [4], developed a detailed 3D FEA model to investigate the FDM process of tool path pattern on residual stress distribution and part distortion. This study concludes that the tool path pattern noticeably affects the part distortions due to residual stress presented on parts. Zhou et al. [5], developed a thermal model on FDM process using ABS material. They used finite element analysis to investigate the temperature variation FDM process based on continuous media theory. In this model they used Element birth/death function used for simulation of FDM process.

The model concluded that the temperature distribution along the ABS filament was almost even. Filip Gorski et al. [6] evaluated the mechanical properties of products manufactured using FDM process and finite element analysis with consideration of strength-affecting process parameters. To simulate the FDM process in a virtual environment model using CATIA V5 CAD and COMSOL, multi physics software was used for calculations. This study concludes that result of bending test and results of FEM computations are compared with experiment and simulation was achieved with a minimum amount of percentage error. Liangbo JI and Tianrui Zhou [7], developed a three-dimensional transient thermal finite element model for Fused Deposition Modeling (FDM) with the help of ANSYS Parametric Design Language (APDL) code to simulate the moving head of nozzle. This model established the temperature fields of FDM process.

Wei Jiang et al. [8] investigated the residual stresses and deformations in direct metal laser SLS process in an integrated thermal and mechanical model. Using finite element software FORTRAN program, with geometry and temperatures imported from a thermal model, the residual stresses and deformations of direct SLS of stainless steel are predicted. This study concludes that the deformation of the vertical direction is caused by shrinkage, while the deformation in horizontal direction is caused by result of thermal loading. Ratnadeep paul et al. [9], developed a 3D thermal deformation model using ANSYS Parametric Design Language APDL14.0 software. The thermal deformation model calculates the shrinkage based on slice thickness, part orientation and material properties.

The deformation model was validated with previous literature. This deformation model allowed practitioners to appropriately select the part orientation and slice thickness that will satisfy the GD&T specification of the part. Denlinger et al. [10], related the process parameters on residual stresses and part distortions in electron beam deposition process. They developed a finite element model for predicting the thermo-mechanical response of Ti-6AL-4V material during the process. A 3D thermo-elasto-plastic analysis is performed to model distortion and residual stress in the work piece. This study concluded with the correlation between measured and computed values for emissivity, stress relaxation and part distortion.

T. Mukherjee et al. [11], developed numerical thermo-mechanical model for prediction of residual stress and distortion in Laser assisted additive manufacturing on important process parameter heat input and layer thickness. In this model three-dimensional, transient heat transfer and fluid flow model is used to accurately calculate transient temperature field for the residual stress and distortion modelling for Inconel 718 and Ti-6Al-4V material. The developed model estimates the appropriate heat input and layer thickness to fabricate dimensionally accurate components. Qiang Chen et al. [12], developed a simulation model for selective laser melting process using a ceramic material. The development of model was based on Beer–Lambert law and level set method.

The model influence of different process parameters on temperature distribution, melt pool profiles and bead shapes and effects of liquid viscosity and surface tension on melt pool dynamics are investigated. 3D simulation model was presented by the scanning strategy of SLM process. Mohamed Omar Ahmed et al. [13], developed a mathematical model to establish the nonlinear relationship between process parameters and dimensional accuracy (Change in length, width and thickness). Using I- optimality criterion technique provides the efficient optimization of FDM process parameters for wide range of factors and levels. Finally, the mathematical models were developed to describe the relationship between input parameters and dimensional accuracy. So many research works have focused on minimizing the dimensional accuracy; thus, establishing effective 3D FEA model to predict the deformation model in FDM process for RP machine users to find deformation of the build part before making the products.

This paper is divided into four sections. Section 2 presents the detailed methodology and explains the development of 3D FEA model. Section 3 explains the experimental setup and section 4 provides the results of the study. Finally, Section. 5 enumerates the conclusions of the study.

2 METHODOLOGY

This section discusses the detailed methodology adopted for addressing various tasks involved in this paper. Total work is divided into three parts. First is development of 3D FEA model, second is experimental work and third one is validation of 3D FEA model with experimental work. Fig. 1 shows the detailed methodology/procedure to predict the residual stress, deformation value and computation of maximum deformation using 3D FEA model and validate them with experimental results of FDM process.

The proposed 3D FEA model was developed using the concept of a one way sequentially coupled thermo-mechanical manner. To mimic the continuous deposition process of FDM, the “Element birth/death” function was used in the ANSYS environment. In simulation process, Acrylonitrile Butadiene Styrene (ABS-P430) material properties were used and for transient analysis, time dependent material properties were considered to produce accurate results. The 3D FEA model was simulated based on Ansys Parametric Design Language (APDL) code [19]. To validate the proposed 3D FEA model, several prototypes were fabricated in a Stratasys Dimension 1200es SST FDM machine. Then fabricated prototypes were measured using Coordinate Measuring Machine (Electronica saphire 464) to find out the part error (change in length, width and thickness) values.

2.1. Development of 3d Finite Element Analysis Model

3D FEA model handles the interaction between transient thermal and static structural fields. When the results of one field analysis provide the loads and boundary conditions for another physics field, the analyses are said to be sequentially coupled. The total analysis was done in a coupled manner i.e. first conducting the transient thermal analysis and then static mechanical analysis. In transient thermal analysis, any part subjected to thermal load and temperature dependent properties of the material was given as an input to this analysis.

The output of this analysis was residual stress and it is due to thermal load. So this analysis calls it as thermal deformation model. The output of thermal deformation model was given as an input for a static mechanical analysis. In this analysis, mechanical properties of the material have been given and arrested the nodes available in the bottom surface of the build product. The output of this analysis was deformation of nodal points. This analysis calls it as structural deformation model. The Fig. 2 explains the coupled field analysis for development of FEA model.

For developing FDM process simulation the commercial ANSYS software was utilized. The simulations were conducted in a one way of coupled thermo-mechanical manner.

The element geometry was chosen based on the ASTM standard specimen, a rectangular parallelepiped had dual attributes (Solid45/Solid70) compatible with the thermo-mechanical analysis and Element birth/death function.

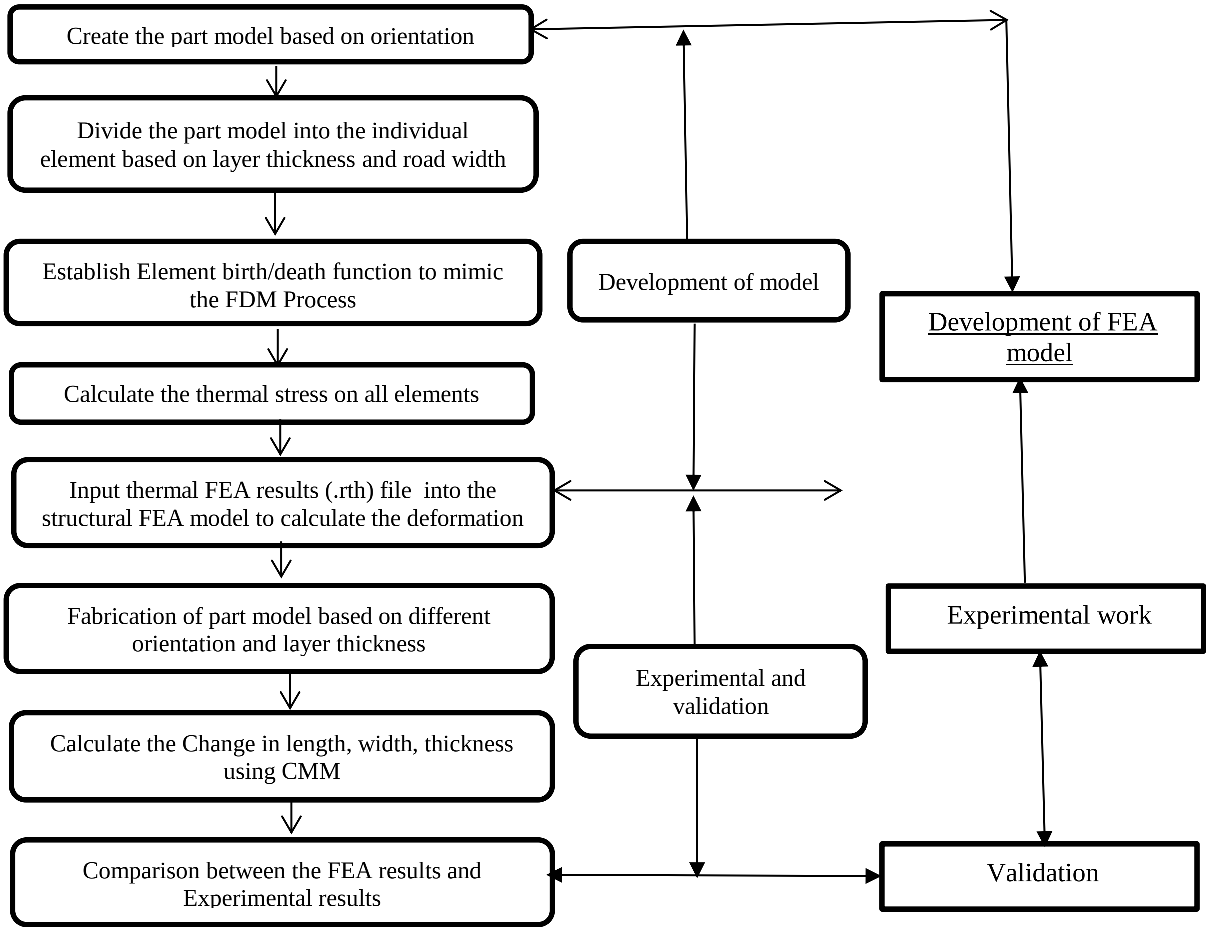


Fig. 1 Detailed methodology of framework

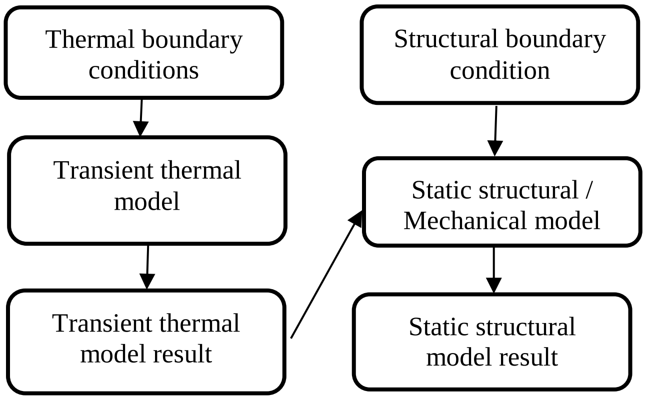


Fig. 2 Coupled field analysis for development of FEA model

The bottom surface of the build product which was in contact with the platform was set to be a constant chamber temperature of 80^0C. The boundary conditions of the outer surface were obtained by forced convection with an ambient temperature of 80^0C. The properties of ABS-P430 plastic and time dependent properties were used in the 3D FEA model. Build material extruded from nozzle was considered as a newly activated element and its temperature was considered as a 300^0C. The temperature of the remaining elements was considered as a temperature of previous element results. The static mechanical analysis was used to find the residual stress and deformation data. In static mechanical analysis the displacement of the newly extruded element was considered as zero and for other elements displacement calculated from the previous results of static mechanical analysis. To mimic the additive feature in FDM process, the novel approach of element birth/death function was used in ANSYS. The model geometry first was specified as per ASTM standard. Then the model geometry was meshed based on the layer thickness and road width, this will lead to formation of elements. The elements were activated according to the filament deposition process equant to tool path pattern followed by FDM machine. The calculation continues until all the elements were activated. Finally plot the graph between residual stress vs deformation of the build part. The 3D FEA model setup was based on orientation, layer thickness and governing equation.

2.2. Governing Equation

In FDM process, the stress/deformation field in a build product would largely depend on the rapid heating and cooling of the build material. Development of 3D FEA model based on governing equation [8], [16].

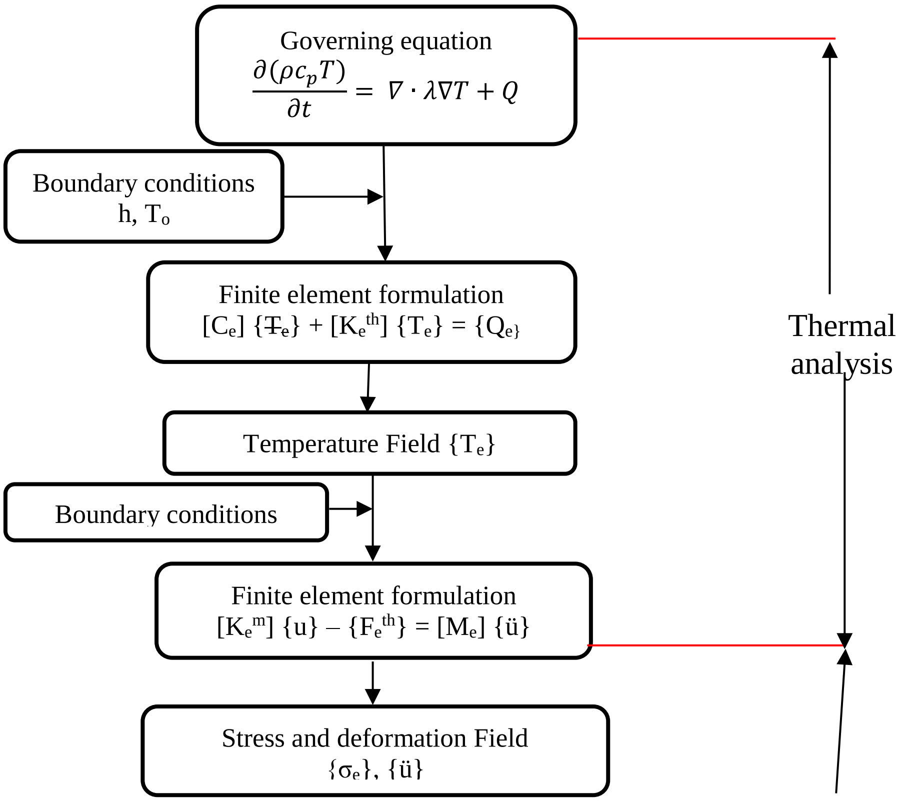


Fig. 3 Governing equation flow diagram for FEA model

Fig. 3 illustrates the workflow diagram for the governing equation flow for FEA model. The thermal analysis quantities {C}, {K} and {Q} represent the capacitance matrix, conductance matrix, and the heat load vector respectively, while the stress analysis quantities {D}, {B}, and {M} represent the stress-strain matrix, shape-function matrix and the mass matrix respectively. The transient temperature field T (x, y, z, t) throughout the process was obtained by 3-D hate conduction equation and same was represented in Eq. (1).

$$\rho c_p(T)\frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(K\frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(K\frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(K\frac{\partial T}{\partial z}\right) + Q$$

$$\text{(or)} \quad \frac{\partial(\rho c_p T)}{\partial t} = \nabla \cdot K \nabla T + Q \qquad (1)$$

Where T is the temperature, ρ is the density, c_p is the specific heat, K is the thermal conductivity, and Q is the internal heat generation per unit volume. All material properties were considered temperature- dependent. The governing Eq. (1) is converted into a finite element formulation which can be written as:

$$[C_e]\ \{Ŧ_e\} + [K_e^{th}]\ \{T_e\} = 0 \qquad (2)$$

Where $[C_e]$ is the element specific heat matrix, $[K_e^{th}]$ is the element diffusion conductivity matrix, $\{T_e\}$ is the nodal temperature in an element, and $\{Ŧ_e\}$ is the change in nodal temperature with respect to time.

The two matrices Eq. (2) can be formulated as based on FEA in ANSYS:

$$[C_e]= \rho\int_v c_p \{N\}\ \{N\}^T dv \qquad (3)$$

$$[K_e] = \rho\int_v c_p [M]^T [J]\ [M]\ dv \qquad (4)$$

Where {N} is the shape element vector, [M] is the shape function derivate matrix, [J] is the material thermal conductivity matrix, and V is the volume of the element. In structural FEA analysis, the material (ABS-P430) was assumed to be perfectly elastic and it is assumed fracture, crack or delamination do not occur in the part. The following Eq. (5) was used in structural analysis. The thermal strain vector for any element within a layer can be written as

$$\{£_{th}\} = (T\ (x, y, z, t) - T_{amb}).[\alpha_L(T)\ \alpha_L(T)\ \alpha_L(T)\ 0\ 0\ 0]^T \qquad (5)$$

Where $\alpha_L(T)$ is the temperature dependent thermal coefficient of the material and T_{amb} is the ambient strain free temperature.

Using Eq. (5) assuming there are no loads on the elements, the nodal displacements can be calculated using the following equation.

$$[K_e^m]\,\{u\} - \{F_e^{th}\} = [M_e]\,\{\ddot{u}\} \qquad (6)$$

Where $[K_e^m]$ is the element stiffness matrix, $[M_e]$ is the element mass matrix, $\{u\} = [u_x\ u_y\ u_z]^T$ is the nodal displacement vector, $\{F_e^{th}\}$ is the thermal load vector, and $\{\ddot{u}\}$ is the acceleration vector.

$$[K_e^m] = \int_v [B]^T [D] [B]\, dv \qquad (7)$$

$$[M_e] = \rho \int_v [N]^T [N]\, dv \qquad (8)$$

$$\{F_e^{th}\} = \int_v [B]^T [D] \{\pounds_{th}\} dv \qquad (9)$$

Where [N] is the element shape function matrix, [B] is the strain, [D] is displacement matrix, and D is the elastic stiffness matrix. Eq. (7) to Eq. (9) are used to calculate the nodal displacements in the part layer elements using the temperature history, T (x, y, z, t), obtained from the thermal analysis.

2.3. Transient Thermal Analysis

Transient thermal analysis was utilized to simulate the heat transfer phenomenon in the presence of time-dependent boundary conditions, body loads, and/or initial conditions along with time-related quantities. Depending on the values of these quantities, solutions to the same problem may differ considerably. Table 1. Shows the thermal properties of ABS-P430 material. Table 2 shows the temperature dependent material properties of ABS-P430 material used in this analysis.

Table 1 Thermal properties of ABS P-430

Sl.No	Property	Units	Value
1	Conductivity	w/m.k	0.16
2	Thermal expansion coefficient	µm/m.K	86

Table 2 Temperature dependent thermal properties of ABS P430

Sl.No	Temperature °C	Specific heat KJ/Kg.K	Enthalpy KJ
1	0	1.62	0
2	105	1.62	109
3	130	3.0	153
4	280	3.0	308

2.3.1. Creation of build product and dividing it into elements

The build product was created using APDL block command and build product dimensions were selected based on ASTM D5418 / ASTM D7028 (13,18) plastic material. The thickness of build product will different for each orientation (i.e) multiplication of layer thickness. The build product dimensions were set to be 35 mm in length, 12.5 mm in width and 3.5 mm in thickness for XY orientation. As a rule of thumb, the solution is expected to be more accurate as the element size is reduced. However, this may increase the cost of analysis significantly. To discretize the deposition process, X direction always set to be 1mm because have to get very accurate result for simulation. In Y direction the element width size set to be based on road width of the FDM process. The value of raster width varies based on nozzle tip size as shown in Fig. 4.

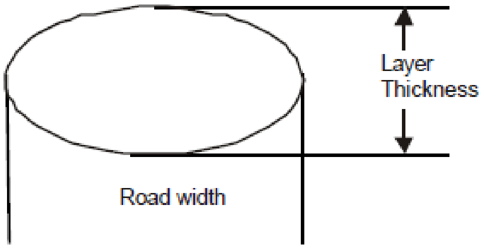


Fig. 4 Road width parameter in FDM process

In Z direction, the element height was set to be a layer thickness of the FDM process. A small element 1 * 0.50 * 0.254 mm^3 and 1 * 0.50 * 0.3302 mm^3 corresponding length, width and height of a single element for all orientation was considered for experimental process. In ANSYS software discretization of elements used the concept of shape hexagonal and mapped meshing. Figs. 5-7 show the meshed model based on orientation and layer thickness of the 3D FEA model.

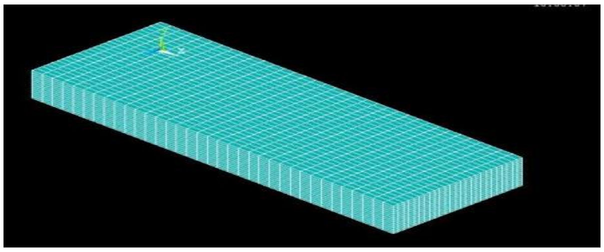


Fig. 5 XY Orientation

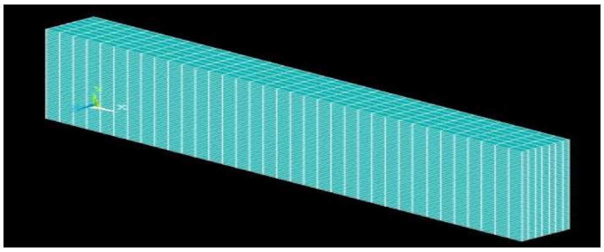


Fig. 6 XZ Orientation

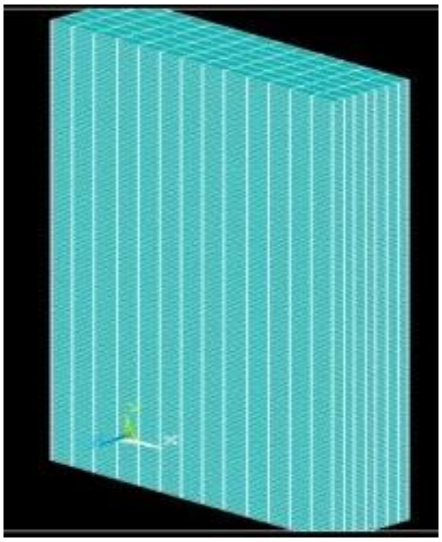


Fig. 7 ZX Orientation

The number of layers and elements were depending on orientation and layer thickness of the build product. In 3D FEA model build product height is based on multiplication of layer thickness. Table 3 shows the number of layers and elements for different layer thickness and orientation.

Table 3 Number of layers and elements for different parameter

SL. No	Layer thickness	Orientation	Number of layers	Number of elements
1	0.2540	XY	14	12250
2	0.2540	XZ	50	12250
3	0.2540	ZX	138	12550
4	0.3302	XY	11	9625
5	0.3302	XZ	38	9310
6	0.3302	ZX	106	9640

2.3.2. Element birth/death function and applying thermal load

The element birth/death function was used in ANSYS [16], to mimic the FDM process. In FDM process, 3D parts were build depositing the material in layered manner one over the other, so that first kill (death) all the elements and then activate (birth) the single element. The method term 'death' does not remove elements to achieve element death, instead of it deactivates them by multiplying their stiffness matrices by a very small value typically 1 * 10^{-9} (ANSYS11). Similarly, when the elements are activated (i.e. 'birth'), their stiffness coefficients return to the original values. The element birth/death technique was used based on the tool path pattern followed by FDM process. To activate the first element and set the bottom surface of the build product was set to be at a constant temperature of 80^0C, heat convection co-efficient of 86 W/m^{2}K and the extrusion head temperature was set to be 300^0C for all nodes on first element. The corresponding deposition time was about 0.031 second per element for giving time for load on the transient thermal analysis. Fig. 8 shows the Load applied condition on first element.

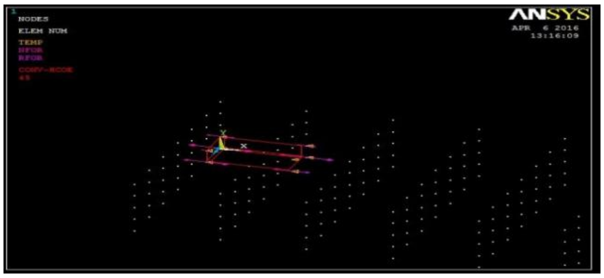


Fig. 8 Load applied condition on first element

2.3.3. Result of Transient thermal analysis

In transient thermal analysis, all thermal results are read on every sub step. In for other elements, initial temperature carries the result of the previous result of transient thermal analysis. Then activate all the elements in the model to produce the temperature field for entire build part. The result of transient thermal analysis is the load of the static mechanical analysis. The Fig. 9 and Fig. 10 show the result of transient thermal analysis of first element and bottom layer.

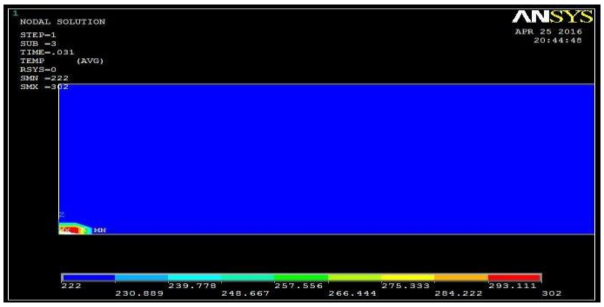


Fig. 9 Result of Transient thermal analysis of First element

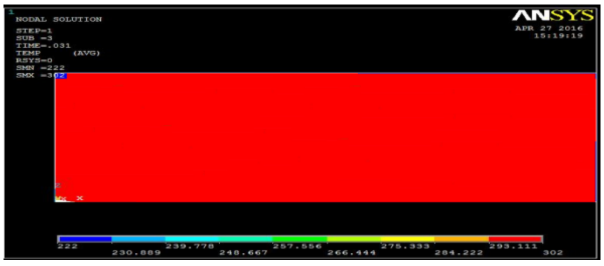


Fig. 10 Result of Transient thermal analysis of Bottom layer

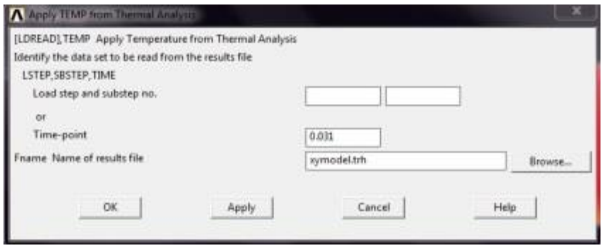
2.4. Static Mechanical Analysis

Static mechanical analysis was utilized to simulate the residual stress and deformation value based on the mechanical boundary conditions, body loads, and/or initial conditions along with the temperature thermal field in the form of (.rth) file gives load to this analysis.

Fig. 11 shows the applied thermal load on static mechanical analysis. Table 4 shows the mechanical properties of the ABS P-430 material.

Table 4 Mechanical properties of ABS P-430

Sl.No	Property	Units	Value
1	Density	kg/m3	1040
2	Poisson's ratio	--	0.36
3	Young's modulus	GPa	2.2


Apply TEMP from Thermal Analysis

[LDREAD],TEMP Apply Temperature from Thermal Analysis

Identify the data set to be read from the results file

LSTEP,SBSTEP,TIME

Load step and substep no.

or

Time-point: 0.031

Fname Name of results file: xymodel.trh

Fig. 11 Apply thermal load on static mechanical analysis

In order to perform static mechanical analysis, the same model geometry was selected from transient thermal analysis. The newly activated element from extruded nozzle was set to be initial as zero displacement at bottom surface and the resulted temperature distribution load has been given as the input of static mechanical analysis. Fig. 12 shows the first element bottom surface nodes of fully arrested.

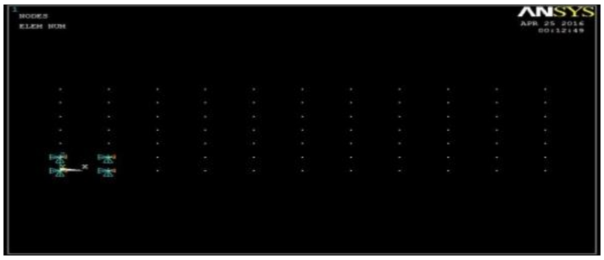


Fig. 12 First element bottom surface nodes of fully arrested

2.4.1. Result of Static mechanical analysis

The proposed 3D FEA model will obtain the maximum principle residual stress and deformation value of a first single element. For other elements, the result of previous static mechanical analysis was used as an initial condition of the new analysis. After activating all the elements, find the total residual stress and deformation of the entire build product. From static mechanical analysis plot the results of residual stress and deformation value. Fig. 13 and Fig. 14 shows the result of static mechanical analysis of first element. The 3D FEA model was performed in ANSYS 11. Nodal list command was used to get the deformation value of each node. Here we mentioned maximum first principal stress and maximum nodal deformation of the entire build product. Table 5 shows the simulation results of principle stress and deformation.

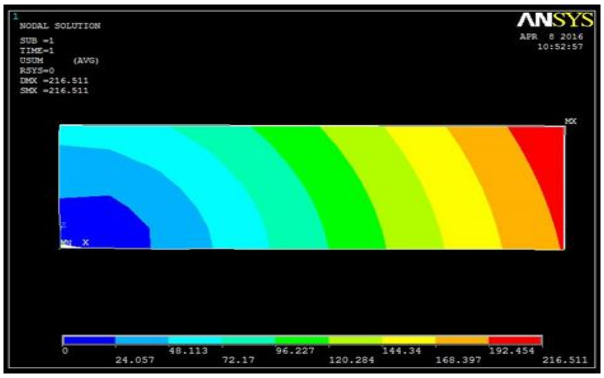


Fig. 13 Result of Static mechanical analysis of 1st element

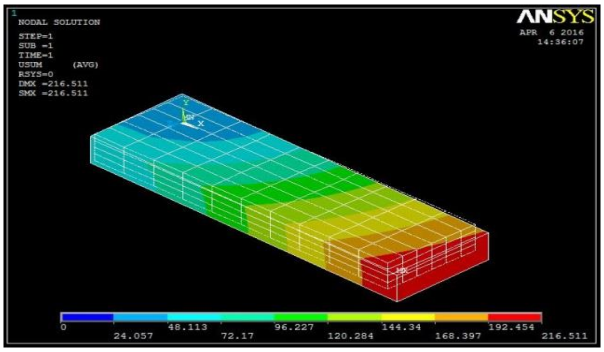


Fig. 14 3D-Result of Static mechanical analysis of 1st element

Table 5 Simulation results of maximum principle stress and deformation

SL. No	Layer thickness	Orientation	Max. Principle stress (Kpa)	Maximum deformation (mm)
1	0.2540	XY	$4.19 * 10^7$	0.2911
2	0.2540	XZ	$2.89 * 10^8$	0.4012
3	0.2540	ZX	$5.67 * 10^8$	0.4532
4	0.3302	XY	$9.14 * 10^7$	0.3790
5	0.3302	XZ	$7.39 * 10^8$	0.4692
6	0.3302	ZX	$8.85 * 10^9$	0.5732

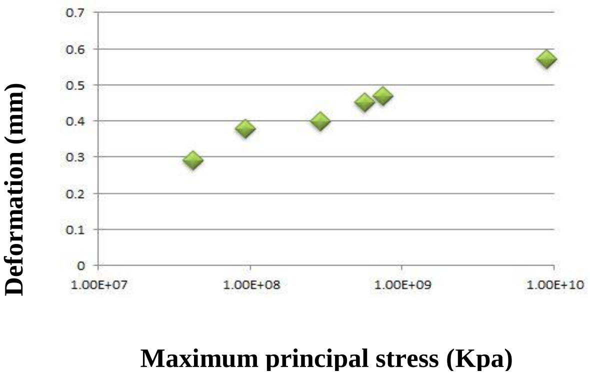


Fig. 15 Graph between maximum principal stress and deformation

Fig. 15 shows the graph between deformation and maximum principal stress. Graph shows that if principal stress increases the deformation of part increase. So optimal process parameter is 0.254 mm layer thickness and XY orientation which gives least deformation compared to others.

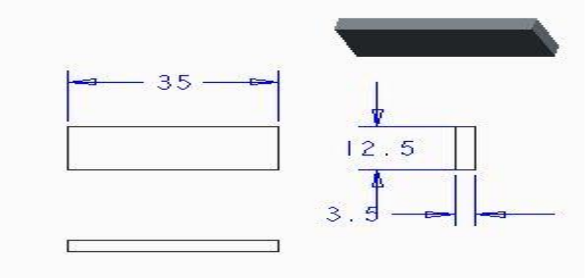
3 EXPERIMENTAL WORK

In this paper two parameters layer thickness and build orientation have been considered. The performance characteristics considered was Part build dimensional error (Change in length, width and thickness). In build orientation we considered three levels such as: XY, XZ and ZX orientation. In layer thickness considered two levels such as: 0.2540mm and 0.3302mm. To find the influence of these parameter on performance characteristics, experiments were conducted in full factorial method (Totally 6 experiments). For each experiments, 3 parts were fabricated in order to avoid the uncertainty in measurements. So totally 18 parts were fabricated using Stratasys Dimension 1200 es SST machine shown in Fig. 16. After fabricated the all the parts, each parts using Electronica saphire 464 CMM machine were measured; to measure the length, width and thickness of the build part. All measurements were taken from Electronica CMM Saphire machine.

Fig. 16 Dimension sst 1200 es FDM machine

3.1. Experimental Setup

The build product were designed based on ASTM D5418 /ASTM D7028(13,18) standared in Creo parametric 2.0 package and saved as a .STL file. The chord height was set as 1 and step size was set as 0 to get high accuracy surface of the part. Fig. 17 shows the build product design using Creo software. Then layer resolution was set to be 0.254 mm and build orientation set to be XY direction, XZ and ZX orientation. Then layer resolution was set to be 0.3302 mm and build orientation set to be perpendicular of each direction. A total of 18 specimens were manufactured using Stratasys Dimension sst 1200es FDM machine.



All Dimensions are in mm

Fig. 17 Part model design

For each experiment, 3 specimens were conducted in order to avoid uncertainty in measurements. The model material used for the test specimen was ABS-P430, with soluble support material. Fig. 18 shows the products of all orientation of 0.3302mm of Layer Thickness.

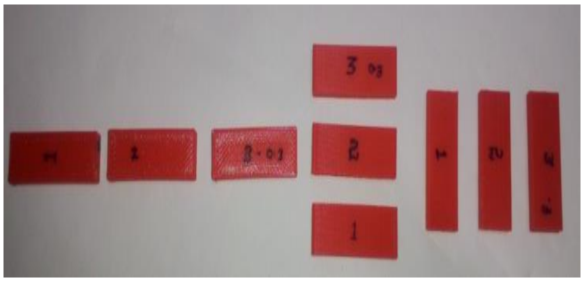


Fig. 18 Build product of all orientation of 0.3302 mm LT

To measure the deformation of the build parts, Electronica saphire 464 CMM machine was used and the accuracy of the machine was 0.5 microns. The raw data points were taken from the build part surface and the reference surface is used for calculating the deformation the electronica made software ABERLINK-3D. It is used to measure the length, width and thickness of the specimen using the point data. For each specimen, 3 sample dimension were collected and then we took average. The Table 6 describes the length, width and thickness of specimen after fabricated by using FDM machine.

Table 6 Final dimension of build part for different factors

SL. No	Layer thickness	Orien tation	Length (mm)	Width (mm)	Thickness (mm)
1	0.2540	XY	35.06	12.48	3.78
2	0.2540	XZ	35.05	12.89	3.54
3	0.2540	ZX	35.43	12.64	3.57
4	0.3302	XY	34.99	12.41	3.86
5	0.3302	XZ	35.18	12.95	3.63
6	0.3302	ZX	35.54	12.64	3.62

The following Table 7 shows the change in length, width and thickness of the specimen from the CAD model. By using this formula: Change in length = Experimental dimension value – CAD dimension value, then all the changes in dimensions are calculated. The Bold caption in change in length, width, thickness was maximum deformation compared to other two values for different orientation and layer thickness.

Table 7 Change in dimension of build part for different factors

SL. No	Layer thickness	Orientation	Change in Length (mm)	Change in Width (mm)	Change in Thickness (mm)
1	0.2540	XY	0.06	-0.02	0.275
2	0.2540	XZ	0.05	0.388	0.04
3	0.2540	ZX	0.43	0.14	0.07
4	0.3302	XY	-0.01	-0.09	0.359
5	0.3302	XZ	0.18	0.446	0.13
6	0.3302	ZX	0.54	0.14	0.12

4 RESULT AND DISCUSSIONS

To validate the 3D FEA model based on the experimental results, the maximum deformation value of nodes on entire part in simulation are calculated. From experiment, the change in length, width and thickness of the specimen using CMM are found. In all orientation, maximum deformation occurs only on layer adding direction. For XY orientation change in thickness is considered, for XZ orientation change in width and for ZX orientation, change in height are considered, as maximum deformation of the part of both simulation and experimental results.

The Table 8 shows the comparison of FEA model results and experimental results in relative percentage error. Relative percentage error = (FEA results – Experimental results) / FEA results. Relative percentage of error comes under 6%. The FEA model can predict the deformation of the build part 3-6% deviation.

Table 8 Validation results

SL. No	Layer thickness	Orientation	FEA Results	Experimental Results	Relative % error
1	0.2540	XY	0.2911	0.275	5.4%
2	0.2540	XZ	0.4012	0.388	3.29%
3	0.2540	ZX	0.4532	0.43	5.1%
4	0.3302	XY	0.3790	0.359	5.27%
5	0.3302	XZ	0.4692	0.446	4.94%
6	0.3302	ZX	0.5732	0.54	5.79%

5 CONCLUSION

In this proposed paper the FDM process simulation has been using a sequentially direct coupled thermo-mechanical FEA model with material properties and boundary conditions in ANSYS 11 package. The FEA model first calculates the temperature gradient history at different layers using a transient thermal analysis and then uses the temperature history to calculate the overall thermal and structural deformation of the part. That incorporates the additive feature and thermomechanical phenomena during the material depositions. The additive feature approach also demonstrates the feasibility of using the element activation/deactivation function to simulate the filament deposition. The temperature dependent properties of ABS-P430 were used for simulation to get the accurate results for transient thermal analysis. Then static mechanical analysis was used to find out maximum principle stress and deformation value of each element and whole build product. The total FEA model was simulated using ANSYS APDL code. The FEA model was used to predict the residual stress and deformation value and it also predicted the influence of process parameter on residual stress and deformation on FDM process. Then FEA model was compared with the experimental results which gives nearly 3-6% error only. So this model can be used to investigate the effects of process parameters such as layer thickness, part orientation and road width on part distortions for process optimization. This will allow practitioners to appropriately select the orientation and slice thickness that will satisfy the design specifications (minimal part error) of the part. The future work of this paper is to enhance the FEA model the APDL code with FORTRAN program which improve the speed of the simulation and to decrease the relative percentage of error, it will further enhance using sequential coupled manner of thermo-mechanical.

6 NOMENCLATURE

3-D	Three Dimension
	Density
ABS	Acrylonitrile Butadiene Styrene
AM	Additive Manufacturing
ASTM	American Society for Testing and Materials
APDL	Ansys Parametric Design Language
BC	Boundary Condition
CAD	Computer Aided Design
CMM	Coordinate Measuring Machine
FDM	Fused Deposition Modeling
FEA	Finite Element Analysis
FEM	Finite Element Model
Kg	Kilogram
KJ	Kilojoule- Energy Unit

Kpa	Kilo-Pascal
LT	Layer Thickness
mm	Millimeter
RP	Rapid Prototyping
SST	Soluble Support Technology
Stl	Stereolithography
Q	Internal heat generation
K	Thermal conductivity
σ_e	Principle stress
ü	Maximum Deformation value

ACKNOWLEDGMENTS

The authors thank the management of Thiagarajar College of Engineering, Madurai for providing the necessary facilities to carry out this work.

REFERENCES

[1] Rahmati, S., Ghadami, F., Process Parameters Optimization to Improve Dimensional Accuracy of Stereolithography Parts, International Journal of Advanced Design and Manufacturing Technology, Vol. 7, No. 1, 2014, pp. 59-65.

[2] Chockalingam, K., Jawahar, N., Chandrasekhar, U., Praveen, J., and Karthic, M., Development of Process Model for Optimal Selection of Process Parameters for Geometric Tolerances and Surface Roughness in Stereolithography, International Journal of Advanced Design and Manufacturing Technology, Vol. 9, No. 3, 2016, pp. 103–113.

[3] Sreedhar, P., Mathikumar Manikandan, C. and Jothi, G., Experimental Investigation of Surface Roughness for Fused Deposition Modeled Part with Different Angular Orientation, International Journal of Advanced Design and Manufacturing Technology, Vol. 5, No. 3, 2012, pp. 21-28.

[4] Zhang Y., Chou. K., A parametric Study of Part Distortions in Fused Deposition Modeling Using Three-Dimensional Finite Element Analysis, Journal of Engineering Manufacturer, 2007, pp. 990.

[5] Zhou, Y., Xiong, G., Nyberg, T., and Liu, D., Temperature Analysis in the Fused Deposition Modeling Process, 3rd International Conference on Information Science and Control Engineering, 2017.

[6] Gorski, F., Kuczko, W., Wichniarek, R., and Hamrol, A., Computation of Mechanical Properties of Parts Manufactured by FDM Using Finite Element Method, 10th International Conference on Soft Computing Models in Industrial and Environmental Applications, 2015, pp. 368.

[7] Liangbo, J. I., Tianruizhou, Finite Element Simulation of Temperature Field in Fused Deposition Modelling, Advanced Materials Research, Vol. 97-101, 2010, pp. 2585-2588.

[8] Jiang, W., Dalgarno, K. W., and Childs, T.H.C., Finite Element Analysis of Residual Stresses and Deformations in Direct Metal SLS Process, Solid Freeform Fabrication Symposium, 2000, pp 340-348.

[9] Paul, R., Anand, S., and Gerner, F., Effect of Thermal Deformation on Part Errors in Metal Powder Based Additive Manufacturing Processes, Journal of manufacturing science and Engineering, Vol. 136, 2014, pp. 031009-1-031009-12.

[10] R Erik, D., Heige, J. C., and Michaleris, P., Residual Stress and Distortion Modeling of Electron Beam Direct Manufacturing Ti-6Al-4V, Journal of Engineering manufacture, 2014, pp. 1-11.

[11] Mukherjee, T., Zhang, W., and DebRoy, T., An Improved Prediction of Residual Stresses and Distortion in Additive Manufacturing, Computational Materials Science Vol. 126, 2017, pp. 360–372.

[12] Chen, Q., Guillemot, G., Gandin, C. A., and Bellet, M., Three-Dimensional Finite Element Thermomechanical Modeling Ofadditive Manufacturing by Selective Laser Melting for Ceramicmaterials, Additive Manufacturing, Vol. 16, 2017, pp. 124–137.

[13] Ahmed, M. O., Masood, S. H., and Bhowmik, J. L., Optimization of Fused Deposition Modeling Process Parameters for Dimensional Accuracy Using I-Optimality Criterion, Measurement, Vol. 8, 2016, pp. 174-196.

[14] Erik R, Irwin, D, and Michaleris, P., Thermomechanical Modeling of Additive Manufacturing Large Parts, Journal of Manufacturing Science and Engineering, Vol. 136, 2014.

[15] Ding, J., Colgrove, P., Mehnen, M., Williams, S., Wang, F., and Sequeira Almeida, P., A Computationally Efficient Finite Element Model of Wire and Arc Additive Manufacture, International journal of Advanced Manufacturing Technology, 2014, pp. 227-236.

[16] Madenci, E., Guven, I., The Finite Element Method and Applications in Engineering Using ANSYS®, Second edition, Springer, 2015.

[17] Stratasys, Dimension 1200 es machine Best Practice: Advanced Z calibration, USA, 2014.

[18] Book of ASTM Standards Part 9 – Plastics, Carbon Black, American Society for Testing and Materials, 1961.

[19] ANSYS, ANSYS 12.1 Theory Reference for the Mechanical APDL and Mechanical Applications, 2009.