World Scientific Research Journal Volume 6 Issue 9, 2020
ISSN: 2472-3703 DOI: 10.6911/WSRJ.202009_6(9).0011

Numerical Simulation of FDM Rapid Prototyping Process based on ANSYS

Xingyu Lu[1, *], Mengyu Xu[2]

[1]College of Mechanical Engineering, Nanjing University of Science and Technology, Nanjing Jiangsu, China.

[2]College of Computing, Nanjing University of Science and Technology Zijin College, Nanjing Jiangsu, China.

Abstract

FDM rapid prototyping technology is a kind of additive manufacturing technology, which has a broad application prospect. However, as a key factor, the precision of modeling limits the development of FDM rapid prototyping technology. In this paper, the finite element analysis software ANSYS is used to simulate the FDM rapid prototyping process through the life and death element technology, and the temperature field and stress field distribution of the forming process are calculated. By analyzing the temperature field and stress field distribution in the forming process, the results show that the warpage and wrinkle in the forming process are mainly caused by the uneven distribution of the temperature field of the molding part. Therefore, it is proposed that improving the temperature field distribution in the molding process is an effective method to improve the accuracy of the molded parts. The numerical simulation experiment in this paper provides a basic analysis model for the further analysis of the influence of different process parameters on the forming accuracy.

Keywords

FDM; ANSYS; Precision of modeling; Temperature field; Stress field.

1. INTRODUCTION

Rapid prototyping technology, commonly known as 3D printing, was born in the late 1980s. It is a high-tech manufacturing technology based on material accumulation method. As a kind of rapid prototyping technology, FDM (Fused Deposition Modeling) was first proposed by American scholar Dr. Scott Crump in 1988. It is based on the processing principle of discrete or stacking. The filamentous material is sent to the nozzle by the wire feeding mechanism, which is heated and melted in the nozzle and ejected from the nozzle with a certain pressure. Under the control of the computer, the nozzle and the worktable move in plane according to the relevant section information to finish the coating of the cross-section layer. When one layer is formed, the nozzle rises or the worktable moves down to the height of a cross-section layer, and then the next layer is coated until the product processing is completed.

Compared with other types of rapid prototyping manufacturing process, FDM has the advantages of low processing cost, wide range of molding materials, fast molding speed, high utilization of raw materials and easy maintenance. However, it is difficult to control the forming process and the molding process parameters, and the molding accuracy is not high. At present, the research on FDM rapid prototyping process is often based on experiments and experience, which does not meet the requirements of high quality and low cost of rapid economic development. The finite element method is used to simulate the whole forming process, and the factors that affect the forming accuracy are found. It can not only improve the research efficiency, but also greatly reduce the research cost.

2. RELATED WORK

Using numerical simulation method instead of a large number of experiments to study the dynamic change law of FDM forming process, which can not only improve the efficiency, but also greatly reduce the research cost.

Xu F, Wong YS and others from National University of Singapore studied the application of genetic algorithm in rapid prototyping. They developed the software system of rapid prototyping process parameters optimization based on genetic algorithm, and gave the detailed algorithm and specific optimization examples [1]. Bellellumeur C et al. established the finite element model of FDM rapid prototyping temperature field based on ANSYS. They simulated the temperature field of FDM rapid prototyping of ABS polymer filament, and obtained the reasonable temperature range [2]. K Thrimurthulu and Pulak M. Pandey proposed a new method to optimize the FDM rapid prototyping process by selecting the appropriate deposition area orientation to improve the surface quality of the molded parts [3]. Kalita SJ, Bose S and others of Washington State University have studied the internal porous characteristics of FDM parts, and analyzed the differences of the holes in different materials after forming, which provided the basis for selecting reasonable materials for FDM [4].

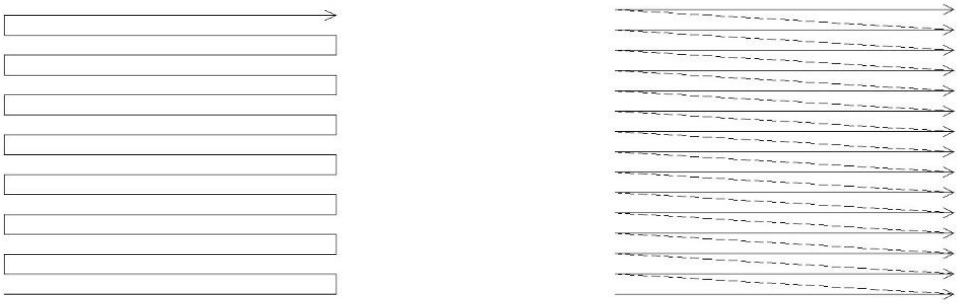
Liangbo Ji of Nanchang University has simulated the temperature field and stress field in the process of melt deposition rapid prototyping. Based on the analysis of the principle and algorithm of artificial neural network, the accuracy prediction subsystem of melt deposition molding product based on wavelet neural network was developed. On this basis, the optimization subsystem based on genetic algorithm was developed, and the software system of melt deposition molding was finally realized and perfected [5]. Based on the experiment research and finite element numerical simulation of the precision of melt deposition rapid prototyping, Ronghua Ni of Shandong University concluded that the errors caused by molding system, discretization of CAD model, delamination thickness, spinneret width and material shrinkage are the main factors affecting the defects of melt deposition products. It provides an important reference for the precision and quality control of melt deposition molding [6]. Jinling Gao of Harbin Institute of Technology simulated the melting deposition process based on ANSYS. Based on the analysis of the simulation results, a ‘Nine-Grid’ heating bed was proposed to prevent the warping of the molding parts [7]. Based on the analysis of the influence of part forming direction on forming accuracy and efficiency, Yuan et al. established a multi-objective optimization model of part forming direction with the minimum volume error, the lowest height in the forming direction and the minimum supporting volume, and designed an intelligent solution algorithm based on the dominant sorting genetic algorithm, so as to realize the part forming and forming efficiency in FDM collaborative optimization of rate [8]. Zhong Ren et al. have carried on the numerical analysis to the melt fluidity and the heat balance in the molding process, optimized the melt parameters of the melt rapid prototyping process [9]. They carried on the numerical simulation to the molding process by using ANSYS and Gauss heat source moving technology, and proposed that the reasonable scanning speed in the molding process can reduce the warping deformation of the molded part [10].

3. NUMERICAL SIMULATION OF TEMPERATURE FIELD IN FDM FORMING PROCESS

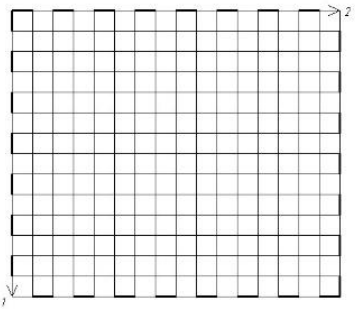
3.1. Numerical Simulation and Analysis Process of Temperature Field

The process of FDM rapid prototyping is a process of continuous accumulation of molten materials, which is a process from nothing to existence. With the continuous increase processing of stacking materials, that is, new heat sources are constantly added, and the temperature field of the whole molding part changes with time. Therefore, the numerical simulation of temperature field of molded parts is a typical nonlinear transient heat transfer process.

The molten material ejected from the nozzle, spread on the worktable according to the prescribed path, and then piled up layer by layer, resulting in the continuous increase of materials in the whole molding process. In the actual processing, the increasing or decreasing of material is reflected in the numerical simulation, which is the ‘growth’ or ‘decline’ of the model unit, that is, the life and death of the element. In the finite element analysis software ANSYS, the life and death function of the element is realized by modifying the element stiffness. When the element is ‘killed’, its stiffness is multiplied by a very small reduction coefficient, so that these characteristic parameters play a very small role in the calculation process, even can be ignored. Similarly, the ‘growth’ of elements is to restore the relevant parameters of the existing elements, so that they can participate in the analysis and calculation again [11]. Therefore, based on the characteristics of the FDM rapid prototyping process, the ‘life and death element’ technology in ANSYS can be used to simulate the material stacking process.



(a) Continuous line scanning (b) Intermittent line scanning



(c) Cyclic interleaving scanning

Figure 1. Scan path

In the process of FDM, the pattern of each layer is filled according to the specified path of the nozzle. The different scanning path has different influence on the temperature field distribution of the molding part. At present, most of the scanning paths used in the literature are continuous line scanning and intermittent line scanning, as shown in Fig.1(a) and Fig.1(b). These two scanning methods are convenient for programming, but the disadvantage is that there are some differences between continuous line scanning, intermittent line scanning and real scanning paths. In the actual molding process, the scanning path is interlaced between layers, so in order to better simulate the real situation, this paper uses ANSYS parametric design language (APDL) to compile a program to simulate the scanning path of cyclic interleaving. The scanning path is shown in Fig.1(c).

In general, the numerical simulation analysis flow of temperature field in FDM forming process is shown in Fig.2.

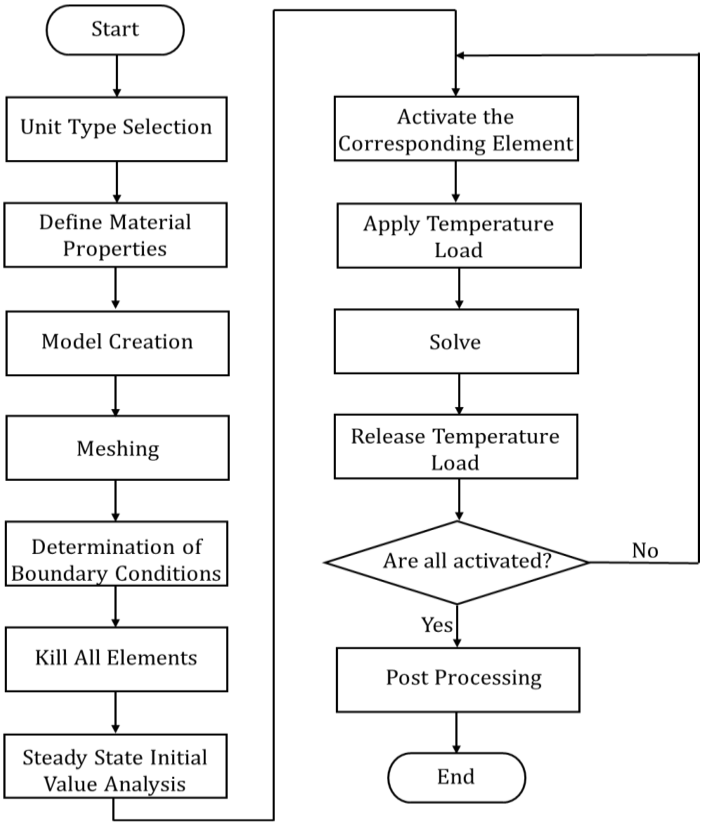


Figure 2. Numerical simulation process of forming parts

3.2. Model Establishment and Load Application

The cuboid of 100mm × 100mm × 4mm is selected as the simulation object, solid70 unit is selected as the analysis unit type, ABS plastic which is widely used in FDM molding process is selected as the molding material, and 2mm × 2mm × 0.4mm is selected as the minimum unit size to mesh the model, as shown in Fig.3.

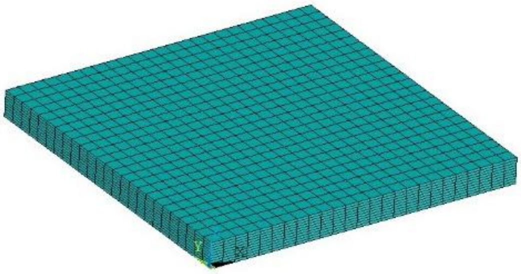
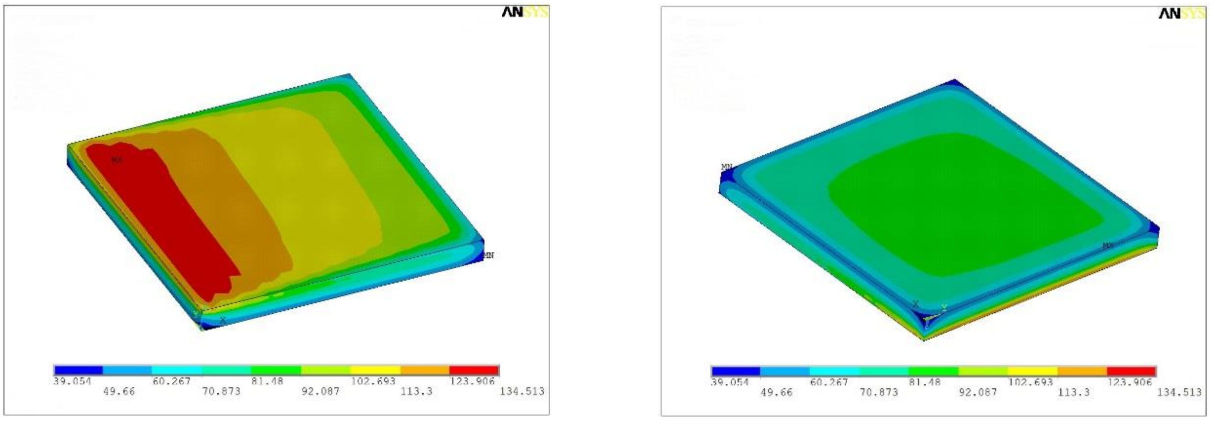


Figure 3. Model Meshing

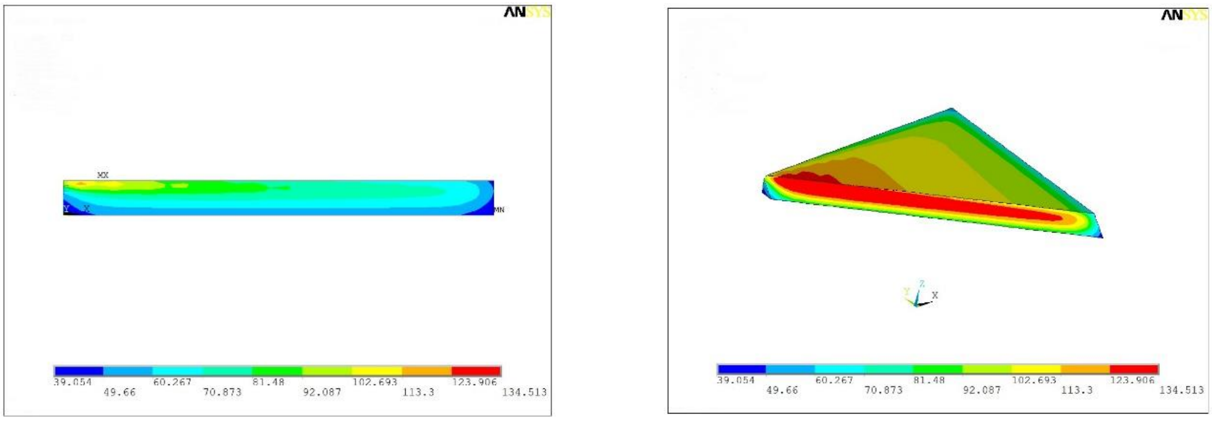
In the numerical simulation of the temperature field of FDM rapid prototyping process, the applied temperature load is 196.415°C, which is the outlet temperature of the nozzle. The boundary condition of molding process is mainly thermal convection, ignoring the influence of thermal radiation.

3.3. Analysis of Numerical Simulation Results of Temperature Field

After solving, the temperature field distribution cloud map of the molded part is shown in Fig.4. It can be seen from the figure that the highest temperature of the molded part is 134.513°C and the minimum temperature is 39.054°C. The temperature of the upper surface of the molded part gradually decreases along the long side from the last scanning track, the lowest temperature appears at the bottom edge, and the temperature in the bottom center area is higher. In the side view, it can be found that the temperature between layers decreases gradually from the upper surface to the bottom surface, which conforms to the basic heat transfer law. However, after the molding part is cut open, the temperature inside the molding part is very high, and the temperature decreases slowly. The temperature field of the whole molding part is generally high temperature in the center and low temperature in the bottom edge. It also shows that the cooling condition in the center of the molded part is poor, and the temperature decreases slowly, while the convective heat transfer condition around the molding part is good, and the temperature drops rapidly.



(a) Top (b) Bottom



(c) Side (d) Section

Figure 4. Cloud Map of Temperature Field Distribution of Molded Parts

On the basis of the temperature field distribution cloud map, in order to further analyze the temperature field distribution, the temperature change curves of the starting node (node = 2) and the bottom central node (node = 376) are selected to print, as shown in Fig.5 and Fig.6. It can be seen from the figure that the starting temperature of node 1 is about 196℃, that is, the temperature of the heat source, and then the temperature drops rapidly. However, due to repeated scanning, the temperature will rise again, but it is still lower than 196℃. Finally, the nozzle temperature is stable at about 40℃. The trend of temperature change with time at node 376 is approximately the same, but the final temperature equilibrium is at 80℃, and the cooling rate is significantly slower than that at node 2.

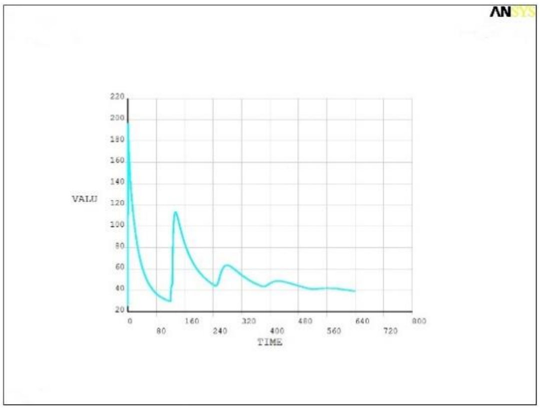


Figure 5. Temperature Versus Time Curve at node = 2

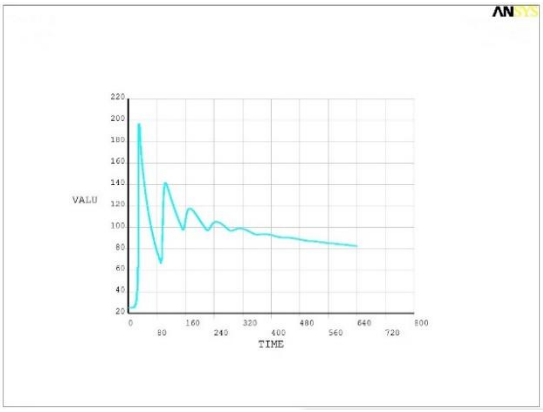


Figure 6. Temperature Versus Time Curve at node = 376

Fig.7 and Fig.8 show the temperature variation curves along the Z-axis direction at the edge and center of the molded part. It can be seen from the figure that the temperature at the edge increases gradually with the increase of the layer height. The minimum temperature is 39.141℃, the maximum temperature is 99.837℃, and the highest temperature occurs at the layer height of 3.6mm. This is because the forming process is a stacking process. The earlier the lower the layer is, the longer the cooling time is, the lower the temperature will be. At the center, the curve is approximately parabola, and the highest temperature appears at the height of 2.8mm. It can be seen that there is a lot of heat left in the molding part, and the heat dissipation condition is worse than the corner.

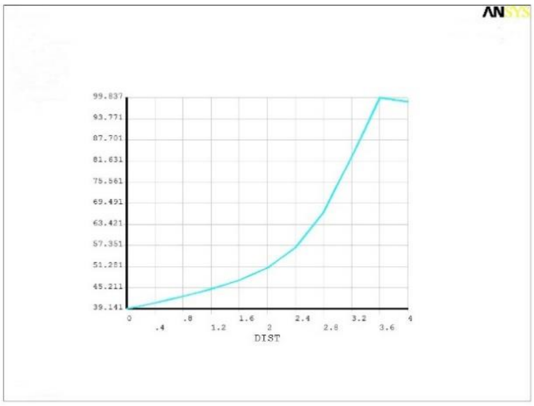


Figure 7. Temperature Change Curve along Z Direction

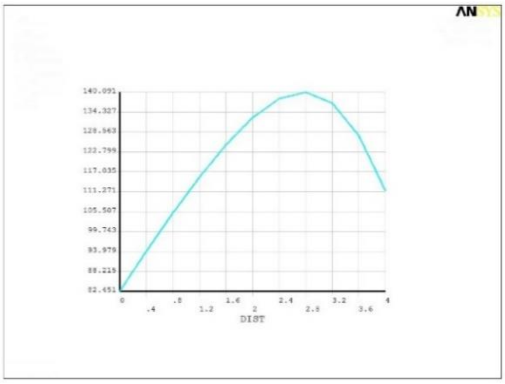


Figure 8. Temperature Change Curve in Z Direction of Center

It can be seen from the above analysis that the heat dissipation condition in the center of the molded part is poor, and the temperature decreases slowly, while the convective heat transfer condition around the molding part is good, and the temperature drops rapidly. As the molding material is ABS, the temperature around the bottom drops rapidly, and ABS material will solidify and harden quickly. At this time, the internal temperature of the molded part is still very high, which will produce great thermal stress and cause the warping deformation of the molded part.

4. NUMERICAL SIMULATION OF STRESS FIELD IN FDM FORMING PROCESS

4.1. Numerical Simulation and Analysis Process of Stress Field

The problem of thermal stress is actually the interaction between thermal and stress fields, so it belongs to coupled field analysis. For such coupled field problems, ANSYS provides two methods to analyze thermal stress, namely direct method and indirect method. The direct method is that the coupled element is used in the analysis, which has two degrees of freedom of temperature and displacement, and the results of thermal analysis and stress analysis can be obtained at the same time. The indirect rule means that the thermal analysis is carried out first, and then the obtained joint temperature is applied to the stress analysis as a load [12]. In this section, the indirect coupling method will be used for analysis, and the thermal analysis unit will be converted into the structural analysis unit, and the numerical simulation results of temperature field will be used as the load of numerical simulation of stress field for corresponding analysis and calculation. The specific analysis process is shown in Fig.9.

4.2. Conversion of Element Type and Application of Load

The thermal analysis unit SOLID70 is transformed into the corresponding structure analysis element SOLID45. SOLID45 element has the characteristics of plasticity, expansion, creep, stress hardening, large deformation and large strain. It is an eight-node hexahedron element. Each node has three degrees of freedom, which are displacements in X, Y and Z directions. It is used to simulate three-dimensional solid structure [7], as shown in Fig.10.

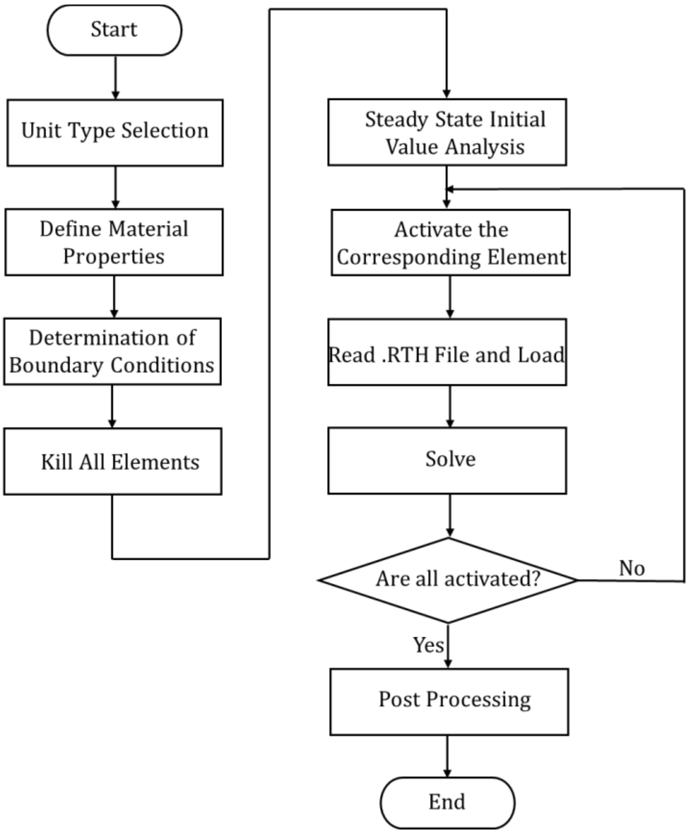


Figure 9. Analysis Process of Stress Field of Forming Parts

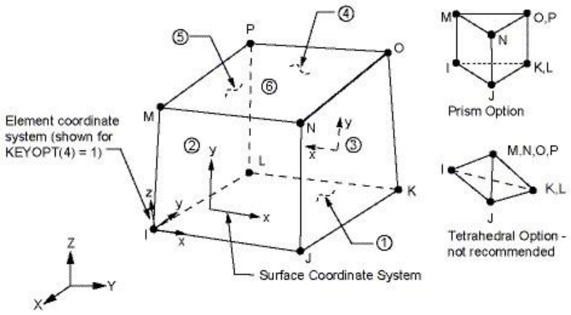


Figure 10. SOLID45 Element

In the simulation process, each element is activated, the corresponding temperature data file is read and loaded, and then the stress field is solved again. This cycle is completed until all the elements are loaded and solved. In the numerical simulation of stress field, the definition of boundary constraint must be carried out. In this analysis, a quarter of the whole molding part is taken for numerical simulation, so the constraint is applied to two adjacent sides, that is, on the cross symmetry plane of the whole forming part, as shown in Fig.11.

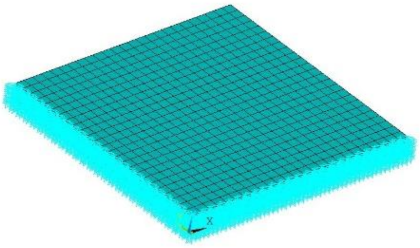
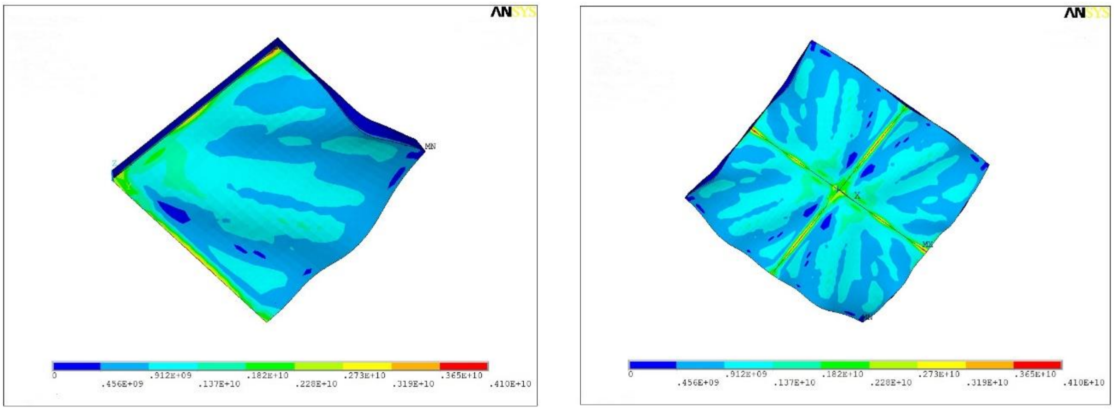


Figure 11. Impose Constraints

4.3. Analysis of Numerical Simulation Results of Stress Field

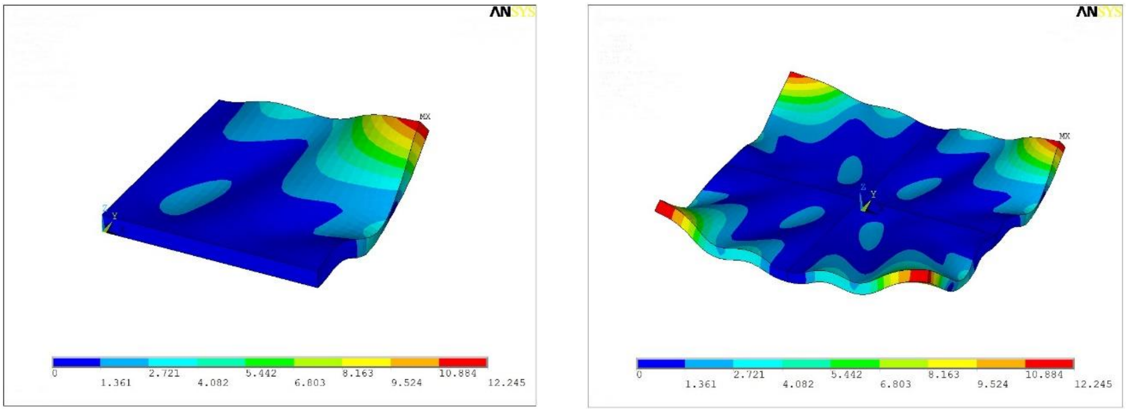
Fig.12 is the stress cloud map of the 1/4 model and the total model at 625th load step. It can be seen from the cloud picture that the stress is mainly concentrated on the cross-symmetry plane at the bottom, and the stress in the center part of the bottom is also large. On the one hand, in the numerical simulation of stress field, the constraint is imposed on the cross-symmetry plane, which makes it difficult to release the stress in this area. However, the stress can be released freely away from the corner of the cross-symmetry plane, which leads to large warping deformation and wrinkles at the corner. On the other hand, combined with the numerical simulation of temperature field, it can be seen that the temperature of the central part drops slowly, while the temperature at the edge decreases rapidly, which will soon solidify and harden, which will also cause great stress in the central region.



(a) 1/4 Model (b) Overall Model

Figure 12. Stress Cloud Map of Molded Part When Step = 625

Fig.13 is a displacement cloud map of the 1/4 model and the total model at 625th load step. It can be seen from the displacement cloud map of the molded part that the main deformation of the molded part is warpage and fold, and the overall performance is a certain shrinkage, which is consistent with the basic law of thermal deformation. The maximum warpage occurs at the top of the corner. It can be seen from the above analysis that the cause of warpage and fold deformation is closely related to the temperature distribution of the edge and center.



(a) 1/4 Model (b) Overall Model

Figure 13. Displacement Cloud Map of Molded Part When Step = 625

5. CONCLUSION

Combined with the numerical simulation of temperature field and stress field in FDM forming process, it can be seen that the main reason of forming part deformation is uneven distribution of temperature field, which means that the temperature drop around is too fast and the temperature drop of center is too slow. In order to control the deformation of forming parts, we should start from the uniform temperature field distribution, which mainly includes two aspects. On the one hand, it needs to slow down the temperature drop around the edge, such as applying a certain temperature to the worktable. On the other hand, it is necessary to improve the heat dissipation condition in the middle of the molding part, such as reducing the filling density of the middle layer appropriately, so as to speed up the internal heat dissipation and balance the temperature field distribution.

In this paper, the finite element analysis software ANSYS is used to simulate the process of FDM rapid prototyping, and the distribution of temperature field and stress field is obtained. From the analysis of heat transfer law and deformation trend, the numerical simulation can better reflect the actual process, which also provides a basic analysis model for further analysis of the influence of different process parameters on the precision of molded parts and the selection of the best working parameters.

REFERENCES

[1] Xu.F, Wong YS, Loh HT: Toward generic models for comparative evaluation and process selection in rapid prototyping and manufacturing, Journal of Manufacturing Systems, Vol. 19(2000) No. 5, p. 283-296.

[2] Bellehumeur C, Li L, Sun Q: Modeling of bond formation between polymer filaments in the fused deposition modeling process, Journal of Manufacturing Processes, Vol. 6(2004) No. 2, p. 170-178.

[3] K.Thrimurthulu, Pulak M.Pandey, N.Venkata Reddy: Optimum part deposition orientation in fused deposition modeling, International Journal of Machine Tool & Manufacture, (2004) No. 44, p. 585-594.

[4] Kalita SJ, Bose S, Hosick HL: Development of controlled porosity polymer-ceramic composite scaffolds via fused deposition modeling, Materials Science and Engineering, (2003) No. 23, p. 611-620.

[5] L.B Ji: Research on the Finite Element Simulation and Process Optimization in Fused Deposition Modeling (Ph.D., Nanchang University, China 2011), p.1-55.

[6] R.H. Ni: Research on Process Precision of Fused Deposition Modeling and Numerical Simulation of Modeling (MS., Shandong University, China 2013), p.1-48.

[7] J.L. Gao: Numerical Simulation of Temperature Field and Stress Field in FDM Rapid Prototyping Machine (MS., Harbin Institute of Technology, China 2014), p.33-58.

[8] Yuan.BB, Jiang ZL: Multi-objective Optimization of the Part Building Orientation in Fused Deposition Molding, Journal of Graphics, Vol. 35(2014) No. 6, p. 835-839.

[9] Ren.C, Huang.J, Jian CL: Research on FDM Technology of Rapid Prototyping for Nozzle Melt, Mechanical Research and Application, Vol. 27(2014) No. 1, p. 62-64.

[10] Ren.C, Dai.Y, Hong.B, Shen ZJ: Research on FDM Process of the Scanning, Journal of Inner Mongolia University of Science and Technology, Vol. 34(2015) No. 1, p. 54-57.

[11] Wen.Z, Zhang WD: Authoritative Guide on Finite Element Analysis with ANSYS 14.0 (China Mechine Press, China 2013).

[12] Zhang CH: Thermal Analysis Course and Example with ANSYS (China Railway Publishing House, China 2007).