

# A New Keyword to Apply Moving Temperature Boundary Conditions and its Application in Fused Filament Fabrication

Jinglin Zheng<sup>1\*</sup>, Danielle Zeng<sup>2</sup>, Xinhai Zhu<sup>1</sup>  
1. Livermore Software Technology, an ANSYS company  
2. Ford Motor Company

## Abstract

*This paper introduces a new keyword, \*BOUNDARY\_TEMPERATURE\_TRAJECTORY, which is capable of applying a prescribed temperature boundary condition within a specified volume which moves along a designated path at a given speed. Combined with LS-DYNA<sup>®</sup>'s computational welding material model, it is able to simulate the process of extruding and depositing melted material onto the build platform, such as the process of fused filament fabrication. An example of printing a dog-bone-shaped plastic part is given to demonstrate the usage of this keyword. The simulation result agrees very well with experimentally measured local temperature histories, which demonstrates the validity and accuracy of this simulation approach.*

## Background

With the rapid development of manufacturing technologies, the demand for numerical modeling of increasingly complicated fabrication processes continues growing. This paper presents a latest keyword development in LS-DYNA, \*BOUNDARY\_TEMPERATURE\_TRAJECTORY, which is designed to meet the rising needs of simulating manufacturing processes involving heat sources with complicated motion patterns. This type of boundary condition, though, can possibly be prescribed with the pre-existing keyword \*BOUNDARY\_TEMPERATURE, the procedure would demand a lot of efforts in script writing from the user side. On the other hand, \*BOUNDARY\_TEMPERATURE\_TRAJECTORY provides an intuitive interface to create and modify such complex boundary conditions with minimum efforts from the user.

This paper briefly introduces the basic features of this keyword. The usage of this keyword is then demonstrated with a concrete example of simulating the process of fused filament fabrication (FFF), a major additive manufacturing method for polymers. In this example, numerical modeling of the material extrusion and deposition along a designated print path is realized by using \*BOUNDARY\_TEMPERATURE\_TRAJECTORY in combination with the computational weld material model \*MAT\_THERMAL\_CWM in LS-DYNA. The thermal analysis result shows excellent agreement with experimentally measured local temperature histories, demonstrating the capability and versatility of this keyword.

## Keyword introduction

The purpose of \*BOUNDARY\_TEMPERATURE\_TRAJECTORY is to apply a temperature boundary condition on nodes enclosed in a user-defined volume, the center of which travels along a designated trajectory at a prescribed velocity. The keyword is composed of three mandatory cards and one optional card, as shown in Table 1. While details of the card structure and meanings of each card entry can be found in the keyword manual of LS-DYNA, some essential capabilities are briefly explained as follows:

1. The user can specify which part or part set that the boundary condition is intended to be applied on (see PID and PTYP in Card 1).
2. The traveling route of the enclosing volume is defined with a nodal path (See NSID1 in Card 1). The nodes that make up the nodal path can be part of the finite element mesh or completely detached from the mesh.

Table 1 Keyword structure of \*BOUNDARY TEMPERATURE TRAJECTORY

	1	2	3	4	5	6	7	8
<b>Card 1</b>	PID	PTYP	NSID1	SPD1	NSID2	SPD2		RELVEL
<b>Card 2</b>	IFORM	LCID	TMULT	LCROT	LCMOV	LCLAT		LCENF
<b>Card 3</b>	P1	P2	P3	P4	P5	P6	P7	P8
<b>Card 4</b>	TX	TY	TZ					

3. Currently, the shape of the enclosing volume can be a standing cylinder, a rectangular block, or a lying tube, as shown in Figure 1. The parameters that define the exact geometry of the volume (P1, P2, etc. as indicated in Figure 1) can be either constants or functions of time (See IFORM in Card 2

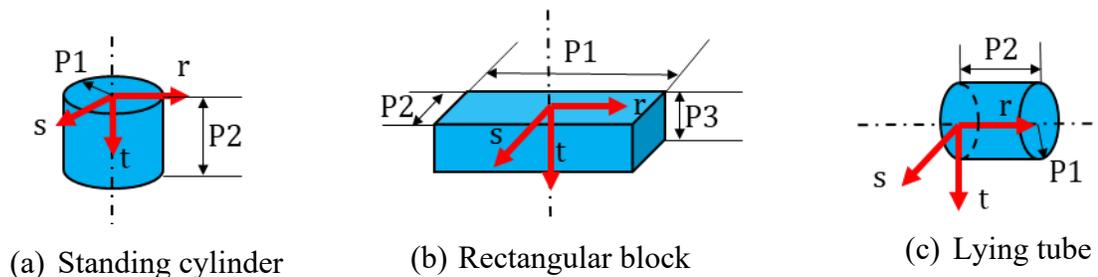


Figure 1 Options of the enclosing volume shape

and P1~P8 in Card 3).

4. With a local coordinate system ( $r$ ,  $s$ ,  $t$ ) (see Figure 1) associated with the moving volume, the volume can be rotated and translated with respect to the traveling path. The amount of rotation and translation can be either a constant or a function of time (See LCROT, LCMOV and LCLAT in Card 2). This allows further flexibility in the boundary condition definition.
5. The prescribed temperature in the traveling volume can be either a constant or a function of time (See LCID and TMULT in card 2).
6. The user can enable or disable this boundary condition at any segment of a traveling path by specifying a time-dependent load curve as the enforcement curve (See LCENF in Card 2). This option comes in handy when simulating, for example, a print head in an idle state (in which case the print head is moving but not depositing any material) in a 3-D printing process.
7. The velocity of the enclosing volume can be a constant or a function of time (see SPD1 in Card 1). And the velocity can be specified as either the absolute velocity or a relative velocity calculated with respect to the nodal path (See RELVEL in Card 1). This option comes in handy when moving parts are involved in the analysis or parts that contain nodes from the nodal path undergo a relatively large deformation.

### Example: Modeling fused filament fabrication of a dog-bone tensile bar

To illustrate how this keyword fits into a practical manufacturing process simulation, here we develop a numerical approach which models the 3D-printing process of a dog-bone tensile bar using FFF. In an FFF process, a polymer filament is continuously fed into a heated print head where the material is melted, extruded

and deposited onto a platform where a part is being formed incrementally. During this process, a temperature field is developed in the growing part and evolves with the movement of the print head. Depending on the process parameters, this temperature field may induce significant warpage in the part, which needs to be minimized or eliminated by improving the fabrication process.

To simulate this temperature field, a couple of physical mechanisms must be characterized in the numerical model: (1) the growth of the part through material deposition; (2) the heating effect on the part caused by the material deposition. In a traditional approach, the material deposition is modeled by activating elements progressively along the print path [1]. In the meantime, temperatures of the newly activated elements are raised to the extrusion temperature accordingly to model the thermal impact. In this approach, the part geometry is discretized with a Voxel mesh, which is composed of rectangular brick elements. A downside associated with this approach is that a large mesh size in general results in a zig-zag pattern at the part boundary [2]. Thus, an excessive number of elements may be generated in the case of a complicated part profile, resulting in an over-consumption of computational resources. On the other hand, the zigzag at the part boundary may also introduce artificial stress concentration into the finite element model and downgrade the numerical accuracy [2].

The current model adopts a different approach: instead of activating elements sequentially, the material deposition is modeled by switching the material status of elements incrementally along the print path. This is realized by using the computational weld material model (\*MAT\_THERMAL\_CWM) for the part to be formed. This particular material can be switched between two states [3]: (1) a quiet state in which the material is assigned with a very small thermal conductivity and a relatively large heat capacity, hence behaves like it is not thermally existent; (2) an active state in which the material is assigned with the polymer thermo-physical properties (taken from reference [4]) hence behaves like a normal polymer material. A temperature range,  $[T_{\text{start}}, T_{\text{end}}]$ , is specified in the material model. The switch from the quiet state to the active state is initiated when an element is heated above  $T_{\text{start}}$  and completed once the element temperature hits  $T_{\text{end}}$ . For an element at a temperature between  $T_{\text{start}}$  and  $T_{\text{end}}$ , a set of material properties that is interpolated between the quiet state values and the active state values is used to dictate the material's behavior at this "limbo" state. It is noted that the switch from quiet to active is one-way. Therefore, the material will remain active even the element temperature drops below  $T_{\text{end}}$  later.

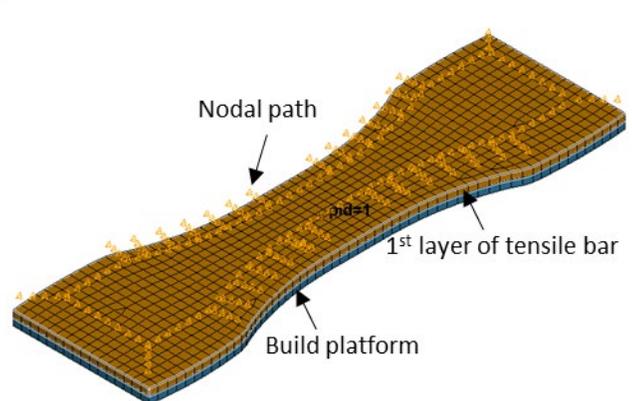


Figure 2 Finite element model of printing a dog-bone tensile bar

Figure 2 illustrates the finite element model used in this approach. The 1<sup>st</sup> layer of a dog-bone tensile part is meshed and seated upon a build platform. The print path is defined by a series of highlighted nodes. The simulation starts with all elements belonging to the part initialized at the chamber temperature, which is lower than  $T_{\text{start}}$ , as shown in Figure 3(a). This leaves the part to be printed at a quiet state, colored in blue, in contrast to the red color of the build platform, as indicated in Figure 3(b). Then the FFF extrusion temperature is applied as a boundary condition along the nodal path shown in Figure 2 using \*BOUNDARY\_TEMPERATURE\_TRAJECTORY. The shape, size and speed of the enclosing volume is specified in accordance with the related process parameters, such as nozzle diameter, infill percentage, infill print speed, etc. This temperature boundary condition forms an evolving temperature field in the part as shown in

Figure 3(c), switching the part material from quiet to active incrementally, indicated by elements turning from blue to red as shown in Figure 3(d). Upon the finish of the 1<sup>st</sup> layer, the finite element model is updated with the addition of 2<sup>nd</sup> layer elements (colored in green), as shown in Figure 4(a). The model is initialized with the temperature field calculated from the 1<sup>st</sup> layer simulation, as shown in Figure 4(b). The same procedure is then continued to calculate the temperature evolution during the fabrication process of the 2<sup>nd</sup> layer. The simulation continues layer by layer until the part is fully finished.

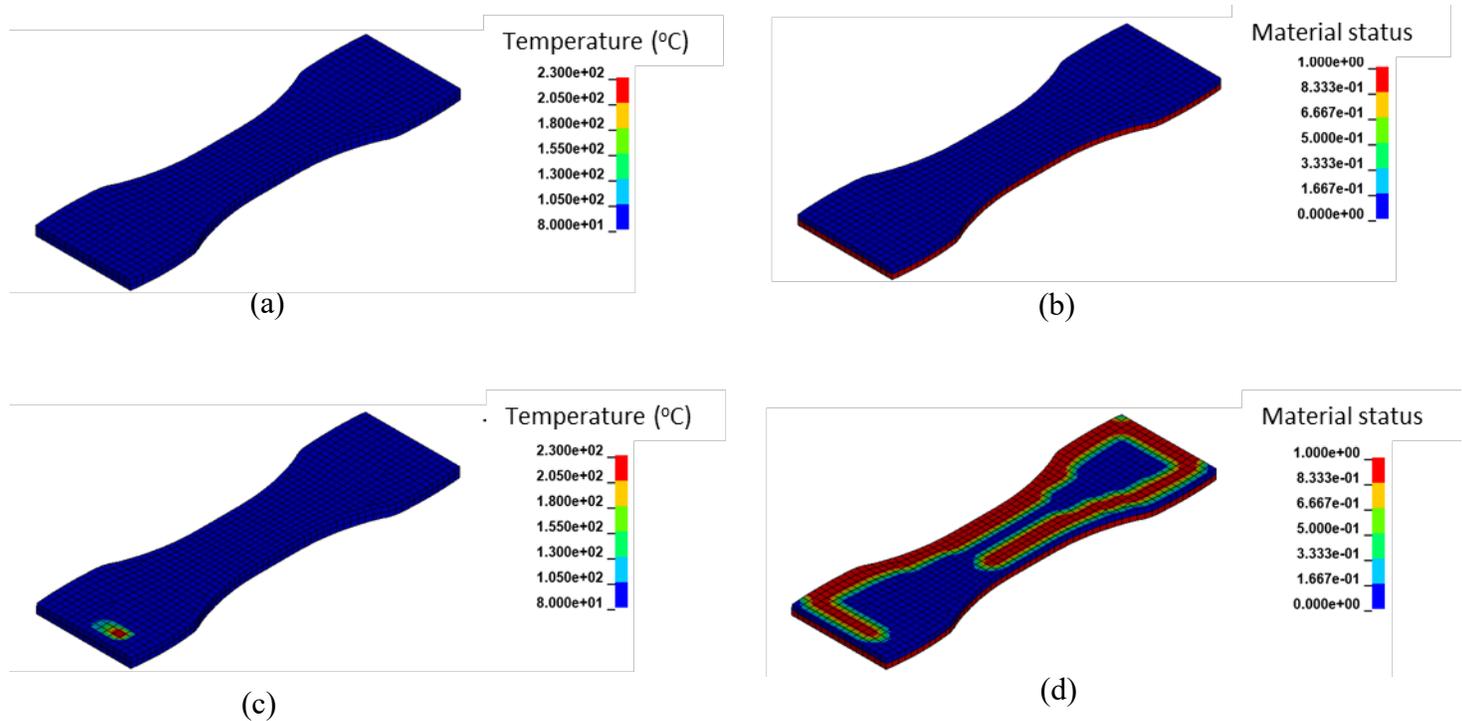


Figure 3 Simulation of the FFF process (a) temperature field at the start of fabrication; (b) material status at the start of fabrication; (c) temperature field during fabrication; (d) material status during fabrication

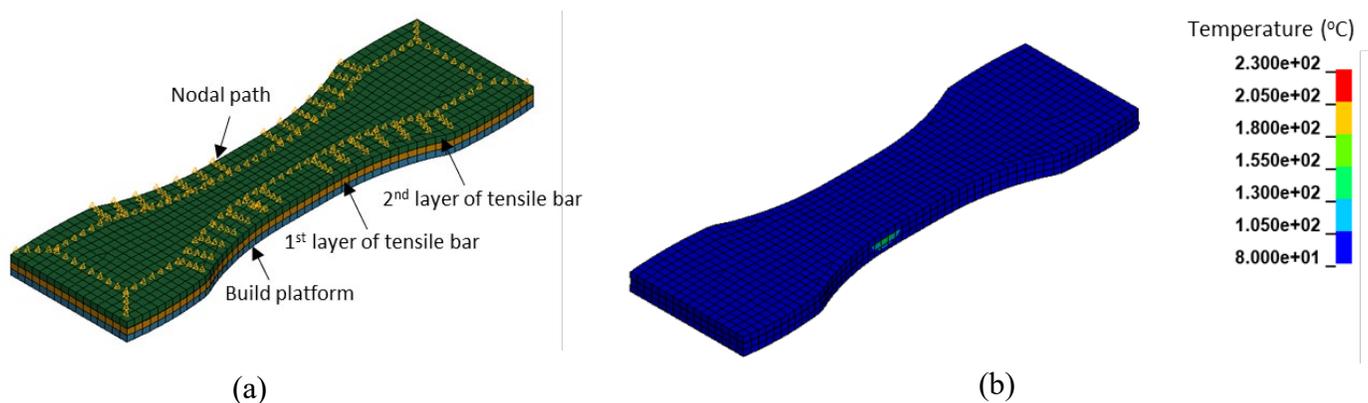


Figure 4 Finite element model of printing the 2<sup>nd</sup> layer of a dog-bone tensile bar (a) updated finite element model with 2<sup>nd</sup> layer elements added; (b) initial temperature field of the model

As illustrated in Figures 2, 3 and 4, this method uses a typical finite element conformal mesh to discretize the part. Therefore, a precise characterization of the part geometry can be accomplished with a moderate number of elements, making the numerical economy and efficient.

## Correlation with experiments

An experimental study <sup>[2]</sup> is carried out to validate the numerical approach presented in the last section. A micro thermocouple is used to measure the local temperature history during the fabrication process. The thermocouple has an extremely low thermal mass to quickly capture local temperature variations and a small enough size to avoid affecting the movement of the print head <sup>[2]</sup>. Figure 5 compares the simulated temperature history and the measured one during the fabrication process of an eight-layer dog-bone tensile bar. It is shown that the local temperature rises when the print head passes by and drops as the print head leaves and the heat dissipates to the surrounding material. Amplitude of the temperature peaks decreases with time because the location of the interested point is located between the build platform and the 1<sup>st</sup> layer of the tensile bar. Thus, the print head, which is the heat source, is moving away from the interested point in the direction normal to the print plane as the print process proceeds. As illustrated by Figure 5, all eight heating and cooling cycles are well captured by the numerical simulation. Based on this thermal analysis, a thermo-mechanical study can be developed in a similar manner to further model the mechanical deformation in the part during the fabrication process. Since an FFF process involves a large number of process parameters, such as the print path, the print head speed, the nozzle size, etc., with the flexible options provided by \*BOUNDARY\_TEMPERATURE\_TRAJECTORY, a series of numerical studies can be carried out to optimize the process settings and improve final quality of the manufactured part.

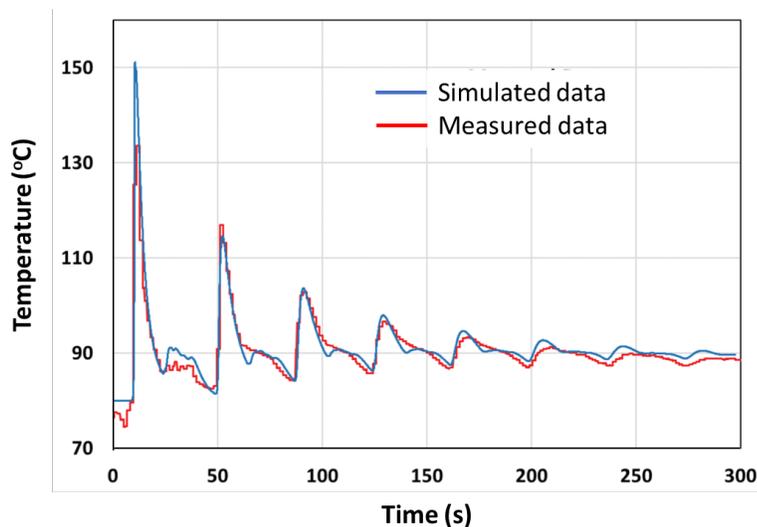


Figure 5 Comparison of simulated and measured local temperature history <sup>[2]</sup>

## Conclusion

A new keyword, \*BOUNDARY\_TEMPERATURE\_TRAJECTORY, is introduced into LS-DYNA, which provides an intuitive and convenient interface to apply a prescribed temperature boundary condition within a user-specified volume which moves along a designated path at a given speed. This keyword finds its application in simulations of manufacturing processes involving complicated heat source movement. As an illustration, a numerical model is developed based on this keyword to simulate the FFF process of a dog-bone tensile bar. Compared to the traditional simulation approach of using a Voxel mesh and activating elements progressively, this model allows the use of a conformal finite element mesh which can capture complex part geometries precisely with a moderate number of elements, leading to a more economy numerical solution. The validity of this solution is confirmed by the excellent correlation between predicted and measured local temperature histories.

**References**

- [1] D. Fradl, J. Panditaratne, J. Bi and V. Oancea, “Finite element simulation of the multi jet fusion (MJF) process using Abaqus”, *Science in the Age of Experience*, 440-469 (2017)
- [2] D. Zeng, M. Rebandt, G. Lacaria, E. Lee, X. Su, J.Zheng and X. Zhu, “Numerical simulation of the temperature history for plastic parts in fused filament fabrication (FFF) process”, *Proceedings of the 30th Annual International Solid Freeform Fabrication Symposium – An Additive Manufacturing Conference*, 1230-1241 (2019)
- [3] LS-DYNA Keyword User’s Manual, Volume II, Material Models (2019)
- [4] F. Tiffaney, M. Gregory, and L. Todd, “Thermal Conductivity Testing Apparatus for 3D Printed Materials”, *ASME 2017 Summer Heat Transfer Conference*, Bellevue, Washington, USA (2017)