

# Numerical Simulation of Temperature Field During Welding of K Type Pipe Joint

Nguyen Hong Thanh<sup>1,2\*</sup>, Nguyen Tien Duong<sup>2</sup>, Ha Xuan Hung<sup>3</sup>

<sup>1</sup>Nam Dinh University of Technology Education, Loc Ha, Nam Dinh, Viet Nam

<sup>2</sup>Hanoi University of Science and Technology, No. 1, Dai Co Viet, Hai Ba Trung, Ha Noi, Viet Nam

<sup>3</sup>University of Labour and Social Affairs, No. 43, Tran Duy Hung, Cau Giay, Ha Noi, Viet Nam

Received: January 07, 2016; Accepted: May 25, 2018

## Abstract

This paper studies the finite element method (FEM) to simulate the temperature field during welding of K type pipe joint. Temperature variations at a point (node) in the heat source movement are examined. Using Gas Metal Arc Welding - GMAW, the welding method is now widely used in the fabrication of steel structure. The modeling and simulation of the temperature field when welding K joint connection made on the basis of the model Solid (3D). In this paper we use VisualMesh / VisualWeld of ESI Group to model and simulate it. This is a useful simulation tool because it can solve the complex problems of thermomechanical and metallurgical in welding including nonlinear problems, pseudo steady state, kinetics, ...

Keywords: GMAW, Temperature field, Piping joint, K Joint, Welding numerical simulation.

## 1. Introduction

Simulation allows shortening research time, reducing research cost and reducing the number of experiments.

Welding simulation is one of the most useful methods to predict the temperature field, residual stress and distortion using FEM. It is widely known that the temperature process directly affects the reaction of the structure in the welding process such as mechanism and microstructure. In the welding process, the welded object is partly heated at a high temperature. According to physical and chemical properties, metals and alloys expand when heated and shrink when cooled. This expansion is limited by areas with lower temperatures or with clamps, which leads to the temporary temperature stress in the welded object and the residual stress after the welded object is cooled. The temperature field identification in welding plays an important role in identifying the residual stress, welding distortion and microstructure of areas in the welding joint.

## 2. Analysis and simulation tool

Among current commercial welding simulation software such as ABAQUS, ANSYS, SYSWELD..., SYSWELD (VisualWeld) by ESI Group is the most powerful and sufficient specialized software in simulating welding and metal temperature treatment processes. The software can solve complex problems related to nonlinear analysis (in heat transfer,

distortion and phase transformation). The software allows modelling and simulating the welding process most sufficiently and practically.

On that basis, the authors choose the software VisualWeld to simulate the structural carbon steel K pipe welding joint in order to study the heat transfer and the formation of residual stress and distortion during GMAW welding.

### 2.1 Heat source model

The heat distribution in the welded object is the heat of the arc column, thermal conductivity of the base metal, the heat release into the environment and the material's thermal properties. With the arc welding heat source, the total of effective power is  $P = \eta \cdot U_h \cdot I_h$  (W), in which  $U_h$  is the arc voltage (V);  $I_h$  is the welding amperage (A) and  $\eta$  is the arc welding efficiency (0.6 ÷ 0.9).

Goldak et al. [1] presented the heat source model in which the double ellipsoid distribution density was defined by the combination of two different semi-ellipsoids to make a heat source (Fig. 1). The heat density in each semi-ellipsoid is described in two separate equations.

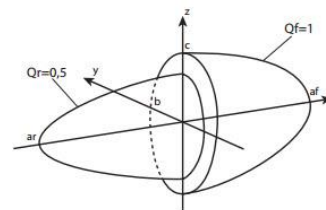


Fig. 1. GMAW welding heat source model, [2]

\* Corresponding author: Tel: (+84) 942283232, E-mail: thanhspknd.we@gmail.com

With any point (x,y,z) inside the first semi-ellipsoid (in front of the welding arc), the heat source density is presented (Eq. 1).

$$Q_R(x, y, z, t) = Q_f \cdot \exp\left(-\frac{x^2}{a_f^2} - \frac{y^2}{b^2} - \frac{z^2}{c^2}\right) \quad (1)$$

With any point (x,y,z) inside the second semi-ellipsoid (behind the welding arc), the heat source density is presented (Eq. 2).

$$Q_R(x, y, z, t) = Q_r \cdot \exp\left(-\frac{x^2}{a_r^2} - \frac{y^2}{b^2} - \frac{z^2}{c^2}\right) \quad (2)$$

where  $Q_f$  and  $Q_r$  represent the maximum power density of the front and rear part of the model respectively and  $a_f$ ,  $a_r$ ,  $b$  and  $c$  represent parameters which enable changing the model geometry and adjusting the shape of the heat source model to results obtained in real welding tests, see Fig. 1

Goldak et al. [3] showed the relationship between the size of the heat source and the size of the welding pool, and supposed that the appropriate values for  $a_f$ ,  $a_r$ ,  $b$  and  $c$  can be achieved by directly measure the geometric parameters of the welding pool. (Table 1).

Table 1. Parameters of moving heat source, [4]

Layers/Pass	b(mm)	c(mm)	a <sub>f</sub> mm	a <sub>r</sub> (mm)
Layer 1 (1, 2)	7	4	3	5
Layer 2 (3, 4)	6	4	3	5
Layer 2 (5, 6)	6	4	3	5

2.2 Material parameters

The material used in K type pipe joint is the structural carbon steel and the material used in simulation is S355J2G3 steel, equivalent to the material used to manufacture K type pipe joint. The chemical composition of S355J2G3 steel are: C (0.18%), Mn (1.6%), Si (0.55%), S (0.035%), P (0.035%) and it has the melting temperature of 1500 °C. S355J2G3 steel has the yield strength of 355 MPa, elastic modulus  $E = 210$  GPa (at 20 °C), poisson coefficient of 0.33. In the welding process simulation, the authors study the temperature variations from 20 °C to the melting status of the material (1500 °C).

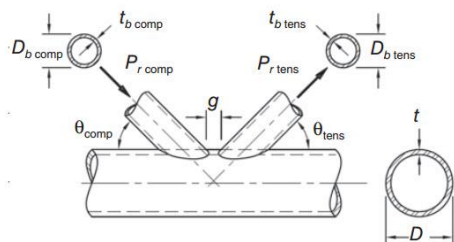


Fig. 2. K type pipe welding joint

Table 2. Parameters of K joint, [5]

Parameter	Description	Unit	Value
D	The outer diameter of the main pipe	mm	219
t	The thickness of the main pipe	mm	12.7
D <sub>b tens</sub> , D <sub>b comp</sub>	The diameter of the branch pipe	mm	102
t <sub>b</sub>	The thickness of the branch pipe	mm	8
θ	Branch angle	°	50
g	gap	mm	50
D/t	Chord wall slenderness	-	17.24
D <sub>b</sub> /t <sub>b</sub>	Branch slender-ness	-	17
D <sub>b</sub> /D	Width ratio	-	0.47
e	Joint eccentricity	mm	0

The size of K joint is designed according to the standard of AISC (Fig. 2). The parameters are described in Table 2.

2.3 Discrete model

In order to properly simulate, it is necessary to describe on FEM model exactly the same as factual welding process, including: welding line trajectory, reference line, start point and end point of the welds. The branch pipe thickness is 8 mm, bevelled edge, we weld 2 layers with 3 passes, see Fig. 3.

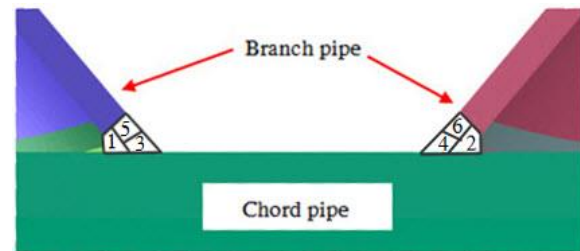


Fig. 3. Welding layers arrangements

The K type pipe joint is discreted model with 145126 elements and 115495 nodes. There are 94288 elements that using in K type pipe joint are hexa, penta, tetra (Fig. 4).

In order to increase the accuracy of analyses, the mesh was concentrated in the joint area as well as in the area adjacent to it (HAZ); in other areas, the distance between nodes is sparser to decrease the calculating time and the number of elements and nodes of the model (Fig. 5).

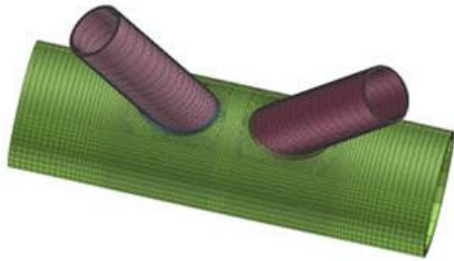


Fig. 4. K type pipe joint modelling

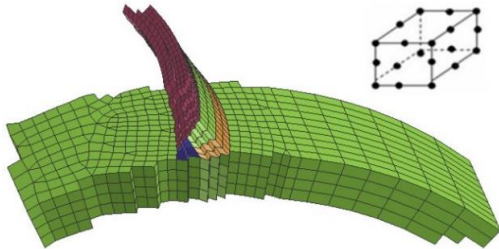


Fig. 5. Type of mesh and mesh dividing area

2.4 Welding process

With K type pipe joint, the welding process and welding layers are arranged as in Fig. 3. To finish this welding joint, we have 12 passes (half a pipe).

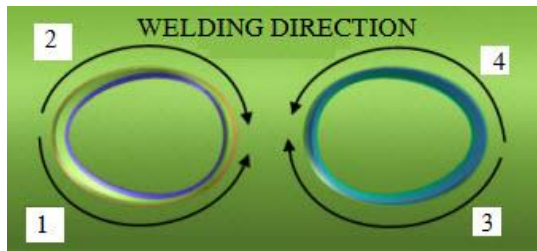


Fig. 6. Welding sequences

In this paper, the author choose the start point of welding line in the narrow angle and the end point in the wide angle. The welding order is respectively 1,2,3,4 as in Fig. 6. The latter welding layers (passes) apply the same as above. The start point and end point of welding passes are alternate.

2.5 The heat source model modification

This is an important step because according to the cross section of the welding joint compared with the temperature field when simulation, we can modify some welding parameters (linear energy, torch angle, parameters  $a_r$ ,  $a_r$ , b and c) suitably to the fusion depth and the width of HAZ area of the weld beads.

Table 3. Welding parameters for GMAW.

Layers/ Pass	$I_h$ (A)	$U_h$ (V)	$V_h$ (mm/s)	$D_d$ (mm)
Layer 1 (1, 2)	120	22	5	1,0
Layer 2 (3, 4)	130	24	5	1,0
Layer 2 (5, 6)	125	23	5	1,0

In which:  $V_h$ : Welding speed [mm/s];  $D_d$ : The diameter of wire [mm].

Table 4. Simulation parameters for GMAW.

Layers/ Passes	Linear energy (J/mm)	Welding speed (mm/s)
Layer 1 (1, 2)	520	5
Layer 2 (3, 4)	610	5
Layer 2 (5, 6)	570	5

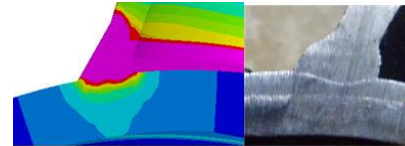


Fig. 7. The heat source calibration

In Fig. 7, the welding mode in Table 4 shows that the fusion depth between simulation and experiment is almost the same and acceptable. Therefore, we can use this heat mode to simulate the temperature of the welding process.

2.6 The clamp condition

The K type pipe joint is clamped as in Fig. 8, in which the main pipe is tightly clamped in three directions (x,y,z) and the two branch pipes are only clamped in the direction z. The clamping time remains during the simulation (4500 s).

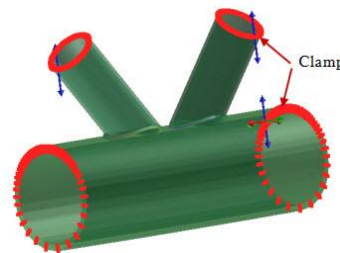


Fig. 8. Clamping position in K type pipe joint

2.7 Calculation and simulation

To get the accurate simulation results, we have to describe parameters exactly the same as the factual welding process, including: welding line trajectory, reference line, start point and end point of the welds (Fig. 9). This process is undergone on the software VisualWeld [6].

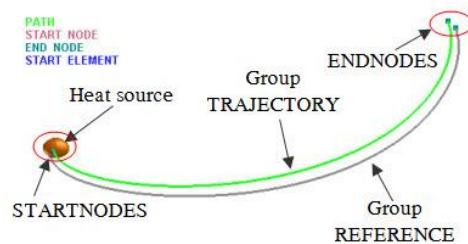


Fig. 9. Weld operation description

### 3. Simulation results and discussion

After modelling the welding joint [7], we declare the properties of materials, the heat source power and set the calculating conditions as described above. Solving the problem, we get the following results:

#### 3.1 The temperature field

When the welding heat source moves along the weld, there appears a temporary temperature distribution field and heat stress in the joint. In Fig. 10 is the temperature distribution results of the welding pool and HAZ area when welding the first line. Fig. 11 is the temperature distribution results of the welding pool and HAZ area when welding the first and the second line.

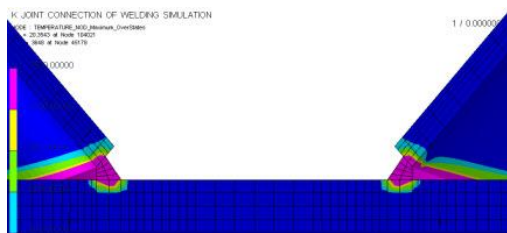


Fig. 10. The total of heat in the first weld

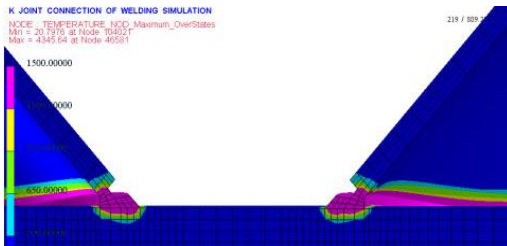


Fig. 11. The total of heat in the first and second weld

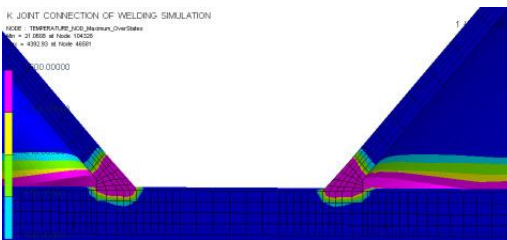


Fig. 12. The total of heat in three welds

Fig. 12 is the temperature distribution results of the welding pool and HAZ area when welding three lines with the heat parameters as in Table 4. Because the heat transfers to the main pipe in 2 dimensions and to the branch pipes in 1 dimension, the heat affected area in the main pipe is narrower than in the branch pipes.

Fig. 13 is the process of heat transfer to the weld, which shows that the heat parameters used in the simulation is appropriate; the joint is not damaged, the heat energy at the beginning and finish of the weld is well treated.

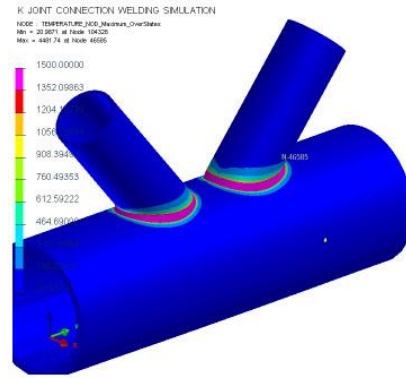


Fig. 13. The quantity of heat to the weld

#### 3.2 The heat process

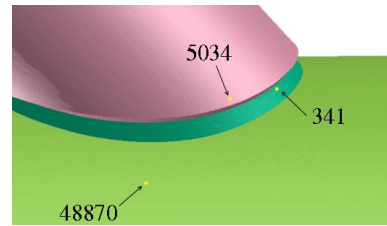


Fig. 14. The total of heat in three welds

Fig. 14 is the location of the nodes defining the temperature field.

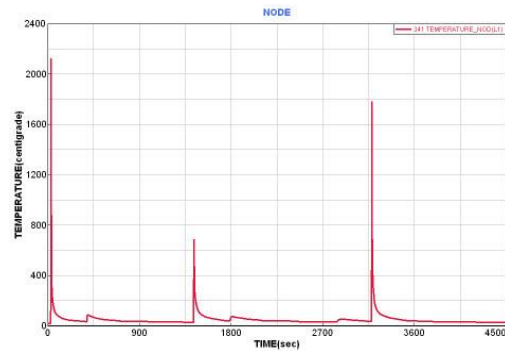


Fig. 15. The heat process at node 341

Fig. 15 is the heat distribution at node 341 in the 1 of the K type pipe welding joint with the useful heat power and the respective welding speed. At 33.1946 seconds, the temperature is 2125.14 °C, this is the time that the melting welding pool pass. After about 317 seconds, the temperature of the joint decrease to 38 °C, at that time, the second bead will be welded.

At the second number 1435.65, the temperature at this node is 685.3 °C; the reason for this increase is affected by the second welding line. However, the second welding line is below this node; therefore, at this time node 341 is affected by the heat as in HAZ area.

It can be seen in Fig. 15 that with the chosen heat source power, the welding joint is not damaged.



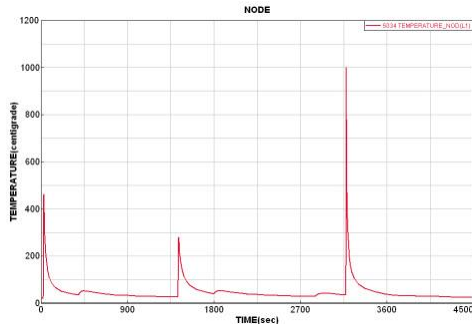


Fig. 16. The heat process at node 5034

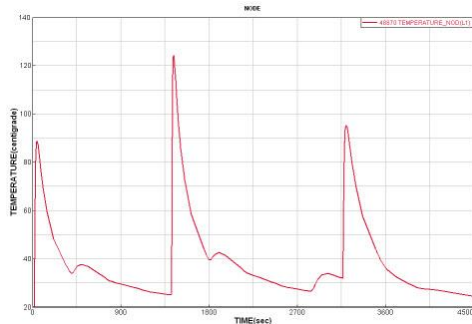


Fig. 17. The heat process at node 48870

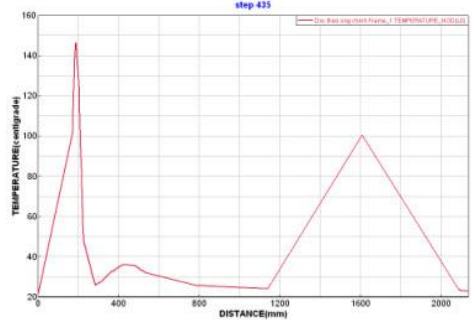


Fig. 18. The heat process along the chord pipe

Fig. 16 is the temperature field at node 5034, this is the node at HAZ area of the branch pipe. This node is heat affected by three welding lines. However, because of the far distance with the second weld bead, the temperature only increases to 300 °C. This temperature does not affect the mechanism and the microstructure of the welding joint and the HAZ area.

Fig. 17 is the heat process at node 48870 in HAZ area and the main pipe is 15 mm away from the weld bead. When the second bead is welded, this node is heat affected (125 °C). This temperature does not affect the microstructure and mechanism in this area.

Fig. 18 shows the temperature variations along the main pipe at step 435 with the highest temperature about 150 °C.

#### 4. Conclusion

This paper focuses to simulate the temperature

field in the K type pipe welding joint with 2 layers and 3 passes using GMAW process. The first pass is the innermost layer, so besides being heat affected itself, it is directly affected by the heat transfer when welding the second and third lines. Therefore, the mechanism and microstructure of the weld happens complicatedly, directly affecting the welding quality.

By changing the input parameters such as the effective heat source power  $P$  (or the linear energy) and the welding speed  $V_h$  using the software Sysweld/ VisualWeld, we can find out the appropriate welding parameter set in order to provide the weld the sufficient quantity of heat to melt the steel but not damage the steel.

According to the temperature field simulation in K type pipe welding joint, we can know:

- + The transfer of the melting welding pool of each welding line and the whole process.
- + HAZ area when welding each line and the whole process.
- + The heat distribution in the chord pipe and the branch pipe.

Numerical simulation is a useful and reliable tool in calculating and analyzing the temperature variation at any time in the welding joint. It can be predicted that the possibility to form the weld and defects can appear during the welding.

Simulation gives rapid results and optimizes the welding parameters to control parameters directly affecting the quality of the weld. We can change the welding parameters rapidly without experiments.

#### References

- [1]. Nguyễn Tiến Dương, Mô phỏng quá trình truyền nhiệt khi hàn, Hà Nội (2008).
- [2]. Nguyễn Hồng Thanh, Hà Xuân Hùng, Nguyễn Tiến Dương, Mô phỏng số trường nhiệt độ khi hàn liên kết chữ T thép A36 bằng phương pháp hàn GMAW, Tạp chí cơ khí Việt Nam, số 6 (2014).
- [3]. Farid Vakili-Tahami. Finite Element Analysis of the In-service-Welding of T Joint Pipe Connections.
- [4]. Eager, T. W., Tsai N. S., Temperature fields produced by traveling distributed heat sources, *Welding Journal*, 62(12). (1983) 346-355.
- [5]. ESI Group, 99 Rue Des, Solets Silic 112 94513 Rungis Cedex FRANCE
- [6]. Goldak, J., Chakravarti, A., Bibby, M., A new finite model for welding heat sources, *Metallurgical Transaction*, 15B (1984) 299-305.
- [7]. Goldak, J., and Akhlaghi, M. (2005) *Computational Welding mechanics*, Springer Science Business Media, Inc.
- [8]. Jeffrey Packer, *Steel Design Guide* (2010) 96-97.