ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XV, NR. 1, 2008, ISSN 1453 - 7397

Adelin Tuţă, Dorel Spiru Dumitriu

Finite Element Analysis of the Polyethylene Pipe Heating during Welding with a Heating Plate

In this paper we present the results of a study based on the finite element method applied to end-to-end polyethylene pipe welding. In the experiment we used the ANSYS 6.1 finite element program that succeeds to accurately capture many common physical phenomena.

Keywords: *polyethylene pipe, end-to-end welding, heating plate,*

1. Introduction

During end-to-end welding with a heating element the pipes made of thermoplastic material undergo a process in which the front ends of the pipes are heated at the welding temperature with a thermal plate. The ends that are heated to the welding temperature are joined by pressing after removing the plate (heating element).

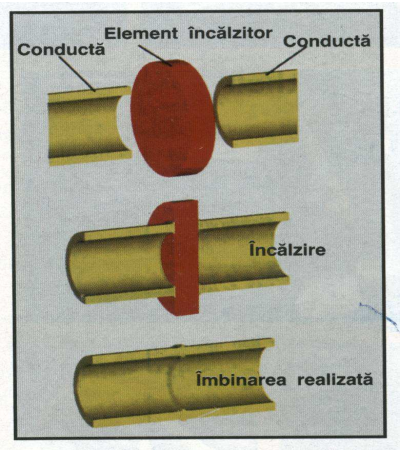

Figure 1. The stages of end-to-end welding with a thermal plate

This technology implies the following phases:

-preheating: this stage, also called adjusting, is defined by the value of the adjusting force. It leads to the formation of the edges at the end of the adjusting. The adjusting force coincides with the welding force. The thermo element is introduced between the ends of the pipes after verifying that the temperature corresponds to the fusion process. The contact between the pipes and the thermal plate creates two separate edges of melted material. The dimension of the edges depends on the diameter of the pipe and its wall thickness;

-heating: this phase is determined by the values of force and time necessary for a complete and homogeneous heating of the contact surfaces of the pipes. The force value generates the contact between the thermo plate and the pipes. At this point the dimension of the edges remains the same as for the adjustment phase. It is compulsory that the passage between the preheating and the heating phases does not include a separation between the heated surfaces of the pipes and the thermo element.

-thermo plate removal: the pipes are separated and the thermo plate is removed.

-welding phase: this phase consists of two steps: firstly the moment in which the pressure increases from zero to the maximum value of the welding force (equal to the adjusting force); secondly the phase in which the force remains constant until the temperature of the welded material reaches 60C. The cooling must be natural without external cooling agents.

During this time interval the weld should not be exposed to any mechanical stress. After this period, the welding process can be considered complete, and the machine can be used in a new process.

In this paper I present the use of the finite element method in the study of end-to-end polyethylene pipe welding. I focus on the first two stages of welding with a heating element.

Nowadays the terms modelling and simulation are understood worldwide to mean the use of computer programs based on the finite element theory. These programs provide accessible and relatively easily interpretable ways of transposing the equations that define phenomena.

The results obtained by using these programs correlate fairly well with reality, especially if the researchers succeed to come close to the essence of the studied phenomena by the approximations and limits of the experiment.

In what follows, more aspects regarding plastic material welding will be studied. These experiments use the finite element program ANSYS 6.1, which accurately renders of many common physical phenomena.

2. The finite element method

This study will analyze the thermal phenomena that take place when polyethylene pipes are welded. In order to obtain satisfactory results more phases

need to be followed, of which the most important ones will be presented.

The first step is to select the mode of analysis, namely thermal analysis. Based on this information the program will offer a series of discretisation elements of the structures which have as degrees of freedom "Temperature". This stage is presented in figure 1. The selected discretisation element is the SOLID 87 type. This is a tetragonal element because the cylindrical geometry of the pipe implies the use of such an element. It has both edges and curved surfaces that overlap the geometrical shape of the analyzed pipe in similar conditions to the real ones.

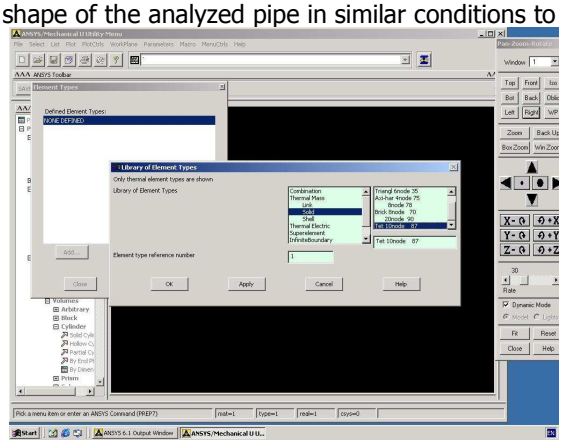

Figure 2. The selection of the discretisation element used: SOLID 87

During the second stage, the thermal and physical properties of the material are input, namely the thermal conductivity, heat and the density of the material. These properties correspond to the material properties and they are entered in the corresponding windows as presented in figure 3.

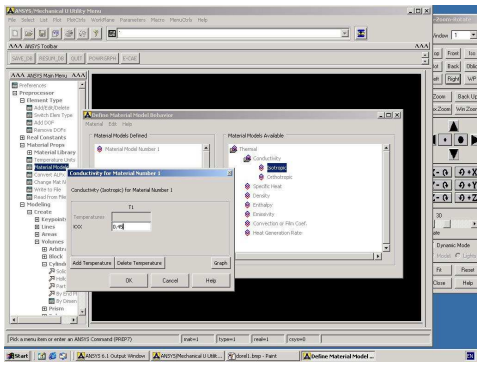

Figure 3. Entering the thermal and physical properties of the studied material

In what follows one of the most important phases is the geometric represen-

tation of the analyzed structure. For the experiment, high density polyethylene pipe with the exterior diameter of 110mm and 10mm wall thickness is constructed. The length of the pipe is assigned at random the value L= 0.2 m. This length is sufficient for thermal analysis since the thermal field does not propagate on this length because of the properties of polyethylene. Figure 4 shows the window that corresponds to the commands needed to build the geometrical construction.

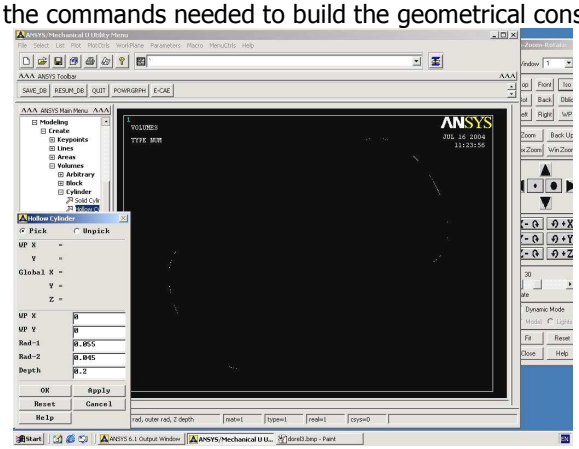

Figure 4. Entering the geometric dimension of the studied structure

In accordance to the already explained commands, the next image shows the analyzed pipe from a thermal perspective during the heating of one of its ends, using a thermo plate with the temperature T=220C.

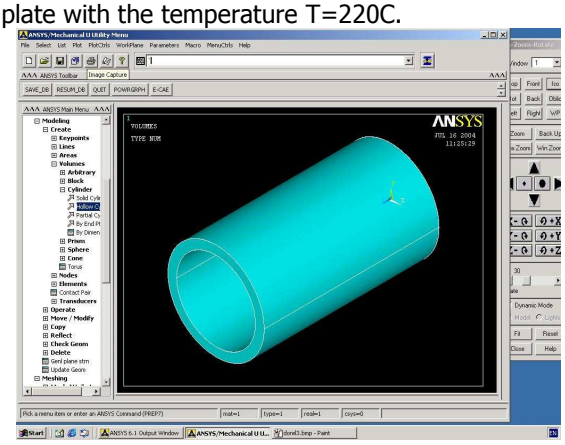

Figure 5. The geometry of the structure

The next phase corresponds to the "mesh generation" of the cylinder by using the discretisation element that has already been presented. Since the geometry of the structure has dimensions of a suitable order of magnitude, mesh generation will use relatively large elements. Therefore it was necessary to set the finest

possible breakdown into component elements in order to ensure that the result of the thermal analysis is sufficiently precise over the maximum possible length of the cylinder generator. The result of mesh generation can be seen in figure 6.

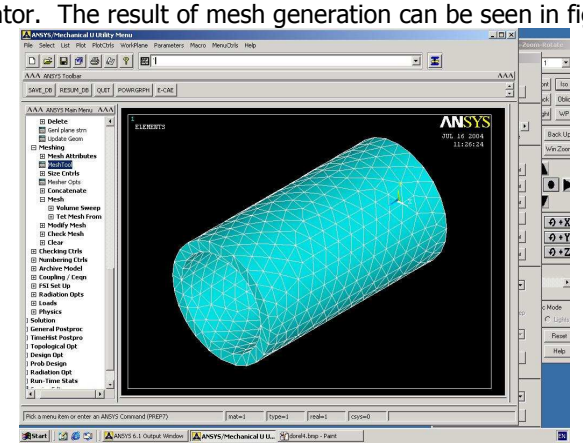

Figure 6. Discretisation of the structure into its component elements

After discretisation, the structure is thermally stressed during the welding operation. The application of the plate at temperature T= 220˚C at one end of the pipe, which has the temperature of 20˚C, represents the real thermal stress that will be modelled with the corresponding commands of the ANSYS program. Figure 7 presents the thermal stressing of the pipe that is heated to T=220˚C at its right end. The generated heat will be transferred to the entire component, simulating the end-to-end welding of two pipes.

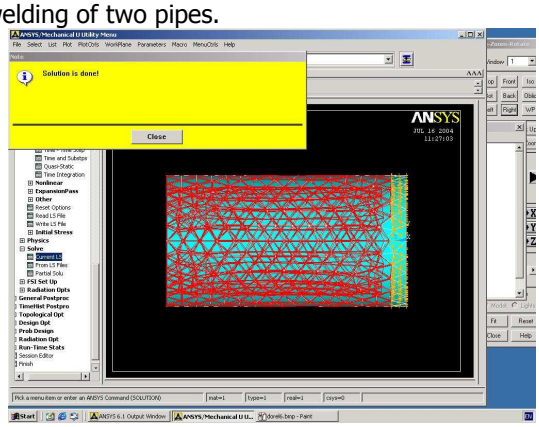

Figure 7. Thermal stress during end-to-end welding

The solution to the thermal problem consists of both the presentation of the thermal field along the pipe and the thermal flux diffusion. Image 8. shows the temperature variation along the generator. As it can be noticed the size of the

surface thermally affected is not big. The pipe is made of plastic material and therefore temperature variation decreases.

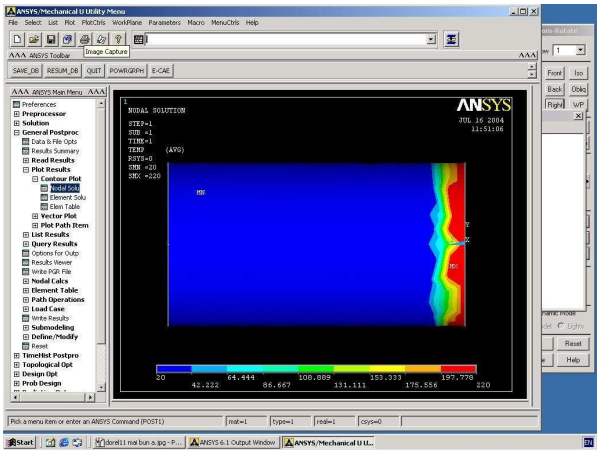

Figure 8. The thermal field during the pressure welding of plastic materials

The thermal flux diffusion in the component is shown in illustration 9. As it can be seen, the thermal flux maximum is not reached exactly at the interface with the heating source, but at a distance a few millimetres.

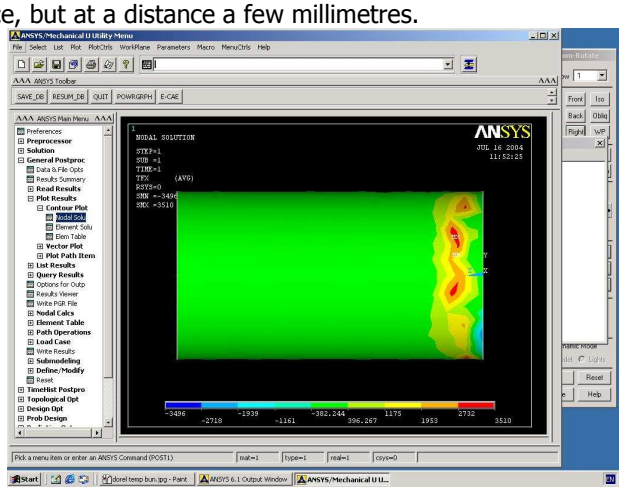

Figure 9. Thermal flux during the plastic material pipe welding

In image 10 the thermal flux is shown in vectorial presentation. The direction of the vectors indicates the thermal field diffusion in the structure of the material and also in the surroundings.

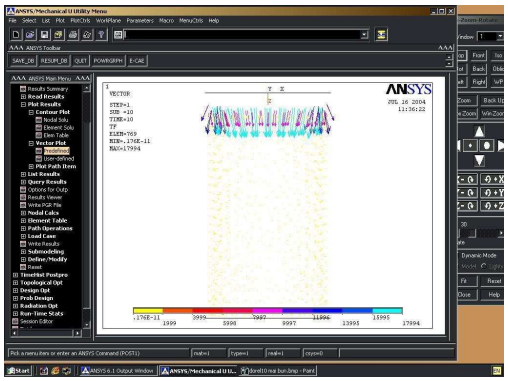

Figure 10. The vectorial representation of thermal flux during end-to-end welding of two plastic pipes

The thermal analysis described shows the conditions in which the welding source constantly heats one of the pipes. It describes the thermal field under these conditions. If the aim is to research the temperature of a node (a certain point), in a specific moment in time, a transient analysis offers this possibility

3. Conclusion

The mathematic modelling of physical phenomena or of certain processes that concern the study of energy is very important. It enables researchers, regardless of the field of study to transpose in an easy manner their ideas and to verify them in a short period of time.

References

[1] Alamoianu, E., *Metoda elementelor finite şi a elementelor de frontieră,* UPB, 1995

[2] Marinescu, G.,Ivan, C., *Metoda elementului finit. Analiza numeric` şi aplicaţii în termoelasticitate*, Editura CIA, Bucure]ti, 1996

[3] Olariu, V., Brătianu, C., *Modelarea numerică cu elemente finite*, Editura Tehnică, Bucureşti,1986.

Addresses:

- Eng. Adelin Tuta, SC EON Gaz Distribuţie Reşiţa, ***adelin.tuta@eon-gaz-distributie.ro***
- Dr.Eng. Dorel Spiru Dumitriu, SC Gospodarul SRL Reşita, dorel_dumitriu@yahoo.com