Numerical Simulation Analysis On Residual Stress Pipeline Girth Welding Joint Strength in Operation Conditions

Peng Zhang[1]**; Linpo Su**[1]**; Xianchun Long**[1] and **Yonghong Duan**[2]

[1]School of Civil Engineering and Architecture, Southwest Petroleum University, Chengdu, China,
[2]Nangchong Heneng Compressed Natural Gas CO.Let, PetroChina Company Limited, Nangchong, China,

Sp. Issue 2013/ # Paper 7
Corresponding author:
Peng Zhang
Mailing address: No. 30 Youyuan Road, Shunqing Zone, Nanchong, Sichuan, 637001, P. R. China
E-mail: zp_swpi@sina.com
Tel : +86-13308229620
Fax : +86-817-2642244

KEYWORDS

Girth seam; Finite element; Residual stress; Strength; temperature

ABSTRACT

After the completion of pipeline girth welding joint, the joint may be strengthened or softened and high residual stress exists. Most of weld indexes are acceptable, but when pipeline works under inner pressure conditions and the weld joint is in the high-accident risk zone. This paper establishes indirect coupling of pipe 3D pipe multilayer V girth weld based on a weld temperature field and stress field of finite element analysis program. The temperature field simulation results are compared with the experimental and simulation results from some references and those Comparison results are analyzed. The results of residual stress strength joint simulation in operation conditions (0MPa, 5MPa, 10MPa, 15MPa inner pressure) are analyzed and their law of variation is summarized.

Introduction

Since the reformation and opening up, China had been formed a polybasic pattern of the supplying energy network. That is “gas from sea landing on, West-to-east gas pipeline, offering in the neighborhood”, and a oil system of “transmission from North to South and from East to West” is built (Ming, 2010), and the west to east gas transmission third pipeline, Ge-La natural gas pipeline, Long-gang pipeline, Shandong natural gas network and from Burma to china natural gas pipeline are being constructed. In addition, most of oil fields have established a large number of gathering pipeline network, where the pipeline diameter is relatively small and the distribution is intensive. Weld is the crucial link of pipeline construction, and it is the foundation to ensure engineering quality. The welding process is a permanent connection between one and the other materials which are the same property or not caused by the binding and diffusion of their atoms or molecules. Girths weld seam is the most common method of connection between pipelines heads. Therefore, the correct understanding of temperature field, stress and strain field when pipeline girth weld is processing or having been processed is the best way to secure those operation and repair them when they are failed, especially conducting design under uncertainty (Zhang, *et al.*, 2010; Wang, *et al.*, 2010; Liu, *et al.*, 2010).

Early code for the assessment of residual stress is based on Pipeline girth weld residual stress measurement values, and the method of residual stress measurement is through-hole drilling, whose drawback is that when stress decreasing gradient is large and the measuring error is large (Dong, 2007).With Ueda SaChio introducing the strength of materials and mechanical properties into the weld thermal elastic plastic finite element analysis theory, weld residual stresses in simulation research which is based on the finite element analysis has achieved rapid development, and finite element used in weld simulation has been widely used. Girth weld simulation of pipeline head can be divided two mayor type: complete circumferential butt welds simulation (Dong, 2001; Mochizuki, 2000; Karlsson and Josefson, 1990) and partial circumferential weld repair simulation (Dong, 2002; Hossain and Truman, 2006).

This article develops a 3-dimension multi-layer steel pipe girth welds thermal and stress program based on APDL, and the last part of stress field analysis program import the different inner pressure block in order to simulation the pipeline operating conditions with residual stress. Based on ANSYS platform runs under these programs, Von mises stress and temperature field of joint have been further analyzed.

Finite Element Models on Girth Weld

Steel pipeline girth weld model is established mainly by several major components of the element selection and material parameters, geometry model, grid division method and the density.

(1) Element Determination

ANSYS thermal analysis program uses SOLID70 element, and the stress analysis program uses SOLID45 element (Dar, 2009). SOLID70 element is eight nodes of the three-dimensional thermal element, and the node exists only one degree of freedom, which is temperature, and it can proceed repeated load coupling, and could realize the 3D steady and transient thermal analysis. SOLID45 is a SOLID70 thermal element corresponding to the stress analysis of converter unit, is also an eight-nodes dimensional unit, which can proceed expanded, elastic-plastic, large deformation and stress analysis in hardening and softening ways.

(2) Material Properties

Indirect coupling welding simulation program of ANSYS is composed of thermal analysis procedure and stress analysis program. Thermal analysis program can realize the pipeline temperature field simulation and it needs to enter material properties as follows: thermal conductivity, specific heat capacity, the convective heat transfer coefficient, Poisson's ratio, density, and the simulated values (Ming and Tang, 1998; Feng, 2006), shown in Table 1. Stress analysis program is a program run by thermal analysis model to simulate the temperature distribution at each step of each node leading-in pipe stress analysis program, and it also needs to enter this program as follows: hot Peng expansion coefficient, elastic modulus after the yield, Poisson's ratio, the yield stress, the simulated values (Ming and Tang, 1998; Feng, 2006) shown in table 2.

Table 1: Thermal Physical Parameters Based on the Thermal Analysis Procedure

* [1]	* [2]	* [3]	* [4]	* [5]	* [6]
20	59	460	1	0.28	7.85
250	43	480	3.5	0.29	7.7
500	35	530	5.2	0.31	7.61
750	33	675	10	0.35	7.55
1000	32	670	15	0.4	7.49
1500	41	660	30	0.45	7.35
1700	45	680	31	0.48	7.3

*[1] *Temperature (degree);* [2] *Thermal Conductivity (W/m.degree);* [3] *Specific Heat Capacity (J/kg.degree);* [4] *Convection Heat Transfer Coefficient (W/m.degree);* [5] *Poisson›s Ratio;* [6] *Density (Kg /m3)*

Table 2 : Thermal Physical Parameters Based on Stress Analysis Program

* [1]	* [2]	* [3]	* [4]	* [5]	* [6]
20	1.1	2.05x10^5	2.05 x10^4	0.28	320
250	1.22	1.87 x10^5	1.87 x10^4	0.29	175
500	1.39	1.5 x10^5	1.5 x10^4	0.31	130
750	1.48	0.7 x10^5	0.7 x10^4	0.35	40
1000	1.34	0.2 x10^5	0.2 x10^4	0.4	25
1500	1.2	0.19x10^5	0.19 x10^4	0.45	0.5
1700	1.1	0.18x10^5	0.18 x10^4	0.48	0.1

*[1] *Temperature (degree);* [2] *Thermal Expansion Coefficient (1/degree);* [3] *Elastic Modulus (MPa);* [4] *Elastic Modulus with Yielded (MPa);* [5] *Poisson Ratio;* [6] *Yield Stress (MPa)*

(3) Establishment of Geometric Model

Petrochemical pipe diameter, pipe materials and wall thickness have many types. For pipe of large diameter and wall thickness, it needs to use large server simulation to achieve adequate precision. In order to get more accurate simulation results, the simulation select pipe diameter of 48mm, wall thickness of 6mm, length of 600mm pipeline are considered as the butt girth welding residual stress and strain analysis object. As shown in Figure 1, pipeline V girth weld inner circumferential weld width of 2mm, outside wall of 8mm, in the welding seam section on 8 special points are used to visually observe the simulation result of temperature field and stress strain field analysis; 1 points and 4 points respectively located inside and outside the center of the weld seam surface; and the distance from point 2 to point 1 is 1mm; point 3 to point 1 is 3mm; point 5 to point 4 is 4mm; and point 6 to point 4 is 6mm..

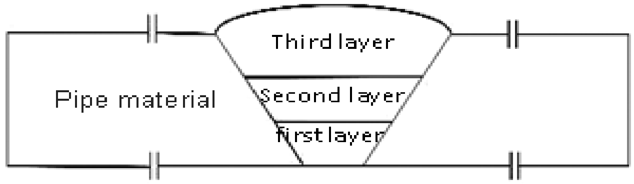

Figure 1: 2D Pipe Girth Weld Seam Model

(4) Grid Division and The Determination of Density

ANSYS software provides free mesh (Free) and mapped mesh (Mapped). Free mesh is applied to two-dimensional models of triangular and quadrilateral mesh partition and irregular three-dimensional entity model of irregular tetrahedral mesh partition. Mapped mesh model is applied to triangular and quadrilateral meshes and three-dimensional entity models of four / five / hexahedral mesh. They all can be identified by element length and grid number and parameter settings to control grid division and meet the requirements of the grid type. Mesh size plays a crucial role on the calculation precision and efficiency, computing speed is slower and time-consuming becomes larger as mesh size become smaller, but the precision of the result has a certain promotion. Thus exterminating the reasonable size of the grid is the first step of welding simulation. Literature (Chen, 2010] on weld unit size where 2mm, and the size of transition zone unit 2mm are used, and we can get better results of temperature field and residual stress strain simulation results from the weld. Distant regions element can gradually increase the length of the element. In order to realize the simulation of multilayer weld, weld along the thickness direction is divided into three layers. The whole mesh partition of model pipeline is shown in Figure 2, and girth weld area is shown in Figure 3.

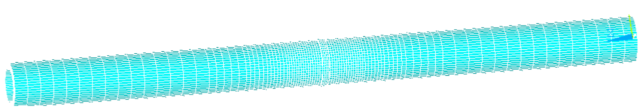

Figure 2: 3D Pipe Model Grid Partition

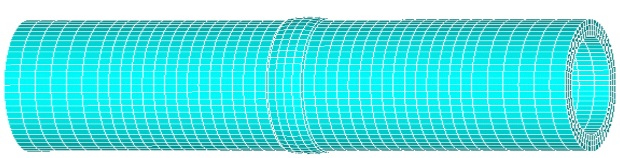

Figure 3: 3D Pipe Model Mesh Weld Area Enlargement

(5) Treatment of Mobile Nonlinear Temperature Load

In this paper, we use indirect coupling simulation, and weld temperature field distribution is simulated, which is the foundation of the simulation of stress field. The weld heat source is the basic conditions to the realization of simulation temperature field distribution. The simulation of thermal analysis uses temperature as the constraint of degrees of freedom, using air convection as the way of heat distribution, the body unit forced on by heat generation rate (Xia, 2007; Wang and Sang, 2011) which is used for energy import mode. Heat generation rate of element is expressed with Q and is given

$$Q = \frac{\eta UI}{V}$$

where η is thermal efficiency; U is electrode and welding between voltage; I is the welding current; and V is the welding speed.

Mobility of heat source is its important characteristic of weld process, and mobile loading of heat generation rate load through the coordinate control element is added in the heat and removing heat and combination to achieve the element birth and death.

Simulative Analysis on Pipeline Girth Welding Temperature Field

In order to make the simulation and pipe girth weld process more closely, this paper adopts multilayer weld simulative technique, and the seam along the thickness direction is divided into three layers using the technique of elemental birth and death. More intuitively display and description of the simulation results is needed, and the unit selects only analysis including weld inside pipes on both sides of 60mm pipeline. The first layer is root weld, and its grid division before the model, shown in figure 4. The second layer is a filler weld, and the grid division before the model, shown in Figure 5. The third layer is cover welding, and the grid partition shown in figure 6. The first step is that the simulation weld process is the first to kill all the weld seam element, and the second step is to determine the direction of welding, shown in Figure

7; 0 to 360 degrees direction through the first weld layer of X, Y, Z coordinate values are successively activated by the first weld layer element and at the time of activation application of heat source is added. The third and fourth step is similar with the second step, which activated the second, third layer welding unit in turn and the simultaneous application of heat source is added. The fifth, sixth and seventh step, get different radiation time within the temperature and stress results (Sddique, 2005).

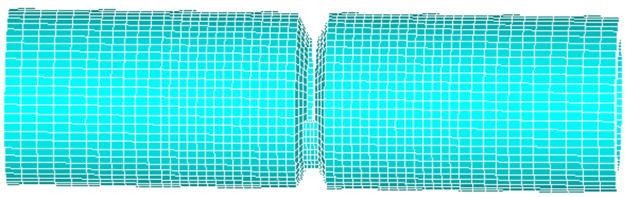

Figure 4: 3D Mesh Model of the Girth Weld First Layer

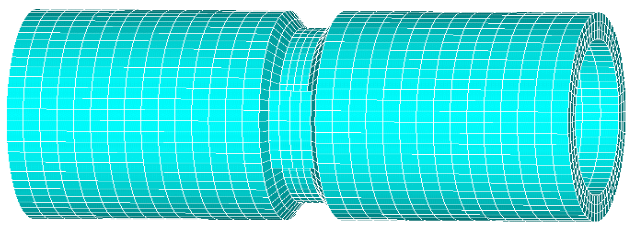

Figure 5: 3D Mesh Model of the Girth Weld Second Layer

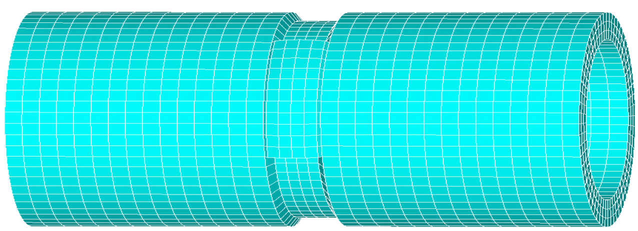

Figure 6: 3D Mesh Model of the Girth Weld Third Layer

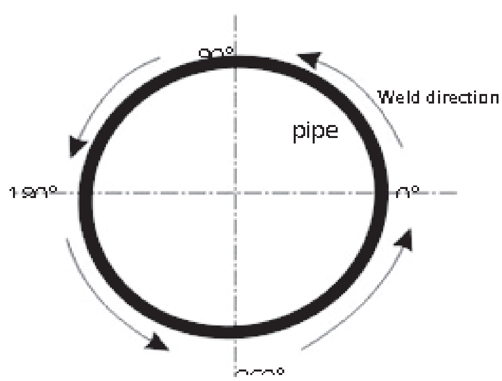

Figure 7: Direction of Simulative Heat Source

Figure 8 shows temperature field when the weld molten pools locate at 90 degree in the first, second and third layer of weld seam center. Clearly first layer 0° weld center temperature has increased four times, and they are the first layer 0° weld time, and the second layer weld time pass first layer 0°, and the third layer weld time pass first layer 0° and weld stop at 360° affecting first layer 0°. The temperature change of the second and third layers are similar to first layer 0°. Instead 180° only increased and reduced three times, because the third layer weld finish on 360° do not affect the 180° weld seam center. From above analysis and result we can conclude that a front layer of heat imports that increase in temperature would affect a latter layer of heat import, and thermal import of a latter layer should meet its materials, so the temperature of the front layer increase again but the maximum of temperature is less than melting point. Because this simulation of pipeline girth weld layers is three, assume weld at *n* layer, and the thermal cycles number of the layer is 3-*n*+1, which is similar with Fig 7 and Fig 8 in reference (Zhao, 2011).

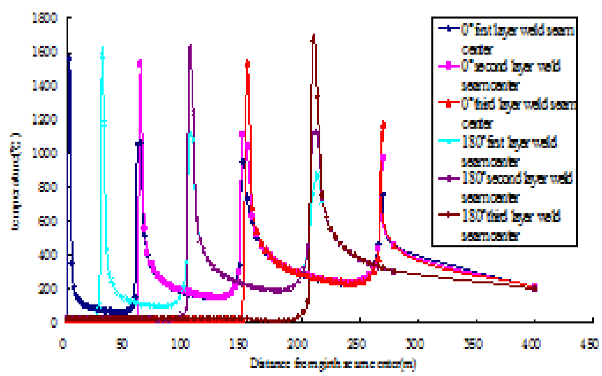

Figure 8: 0°and 180° Weld Seam center Temperature at Different Time Step from the Weld Start

Figure 9 shows the weld molten pool which locates at 90 degree weld seam with six special points that specify temperature changes in Figure 1. Point 1, 2, 3 locate in girth welds and pipes within the wall, and Point 4, 5, 6 locate in girth welds and pipe wall. Heat source above the first layer weld point 1 is reached its highest temperature 1563 °C and heat source above the second layer weld 1 point is reached its highest temperature 1064°C, and heat source above the third layer weld 1 point is reached its highest temperature 958°C, and so accompanied by a welding temperature, highest value is diminishing, and the trend is the same as point 2 and 3. On the contrary, heat source above the first layer weld point 5 and 6 reaches its highest temperature 234°C and 194°C, and heat source above the second layer weld point 5 and 6 reaches its highest temperature 515°C and 415°C, and heat source above the third layer weld point 5 and 6 reaches its highest temperature 996°C and 562°C. From the above analysis, a law can be concluded that the temperature of the inside pipeline and weld seam reduced with the progress of time, but the temperature of the outside pipeline and weld seam has been first increased and then decreased with the progress of time, and the trend is similar with Figures14 and15 in reference (Feli, 2012) and Figure 5 (a-b) in reference (Jiang, 2005).

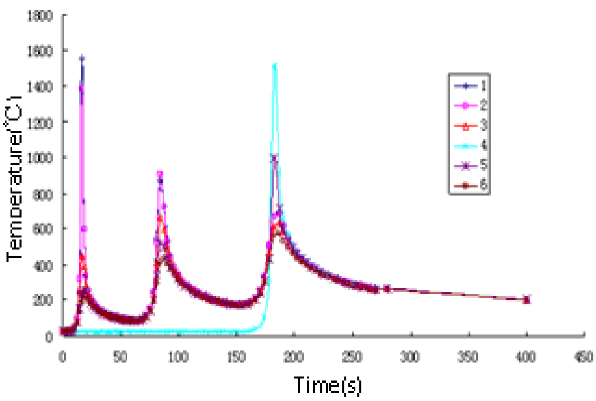

Figure 9: 90°Six Special Pointstemperature at different time step from the Weld Start

Figure 10 shows inner surface temperature of 270° pipe. In Figure 10, curve containing the highest point represents pipeline axis temperature distribution when the molten pool located on the first layer 270° pipe. Curve containing the second high point represents pipeline axis temperature distribution when the molten pool located on the second layer 270° pipe. Curve containing the third high point represents pipeline axis temperature distribution when the molten pool located on the third layer 270° pipe, and the others curves represents pipeline axis temperature distribution when air convection heat sink.

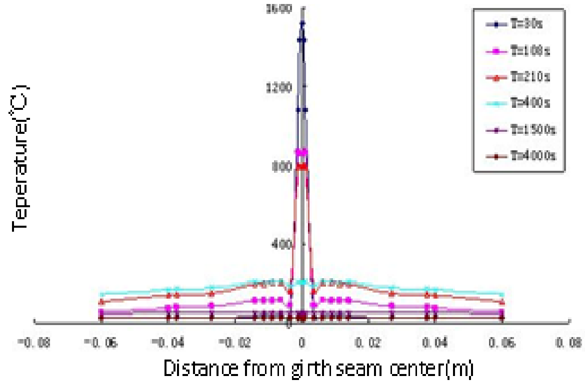

Figure 10: 270° Pipe Weld Head Inner Surface Axial Temperature Distribution at Different Time Step from the Weld Start

Figure 11 shows outer surface temperature of 270° pipe. Figure 11 is similar to Figure 10, and their difference is differential surface. The changing law in the two picture is similar to Figure 10 (a-d) reference (Dar, 2009).

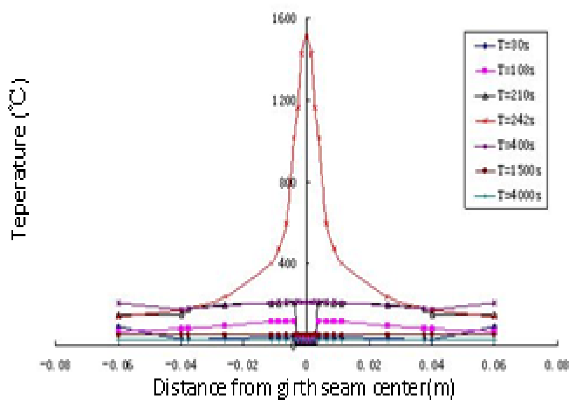

Figure 11: 270° Pipe Weld Head Outer Surface Axial Temperature Distribution at Different Time Step from the Weld Start

Analyze the strength of pipe girth weld joints with residual stress under different inner pressure

Based on no inner pressure in ANSYS welding model, stress analysis APDL programs increase the inner pressure loading blocks. In this way can get the pipe stress analysis model with inner pressure of pipeline girth weld with residual stress procedures, no inner pressure and inner pressure of 5MPa, 10MPa and 15MPa three different pressure levels of strength analysis of pipeline girth welds with residual stress, and explain the result Von mises stress distribution in pipeline girth welds head.

Figure 12 is non-residual stresses, pipeline under 10MPa uniform inner pressure effect of Von mises stress distribution. The girth seam is lowest because it increases the outer metal. Figure 13 with residual stress conditions, Von mises stress distribution imposed 10MPa uniform inner stress, which reflects the basic theory on super positional principle of residual stresses.

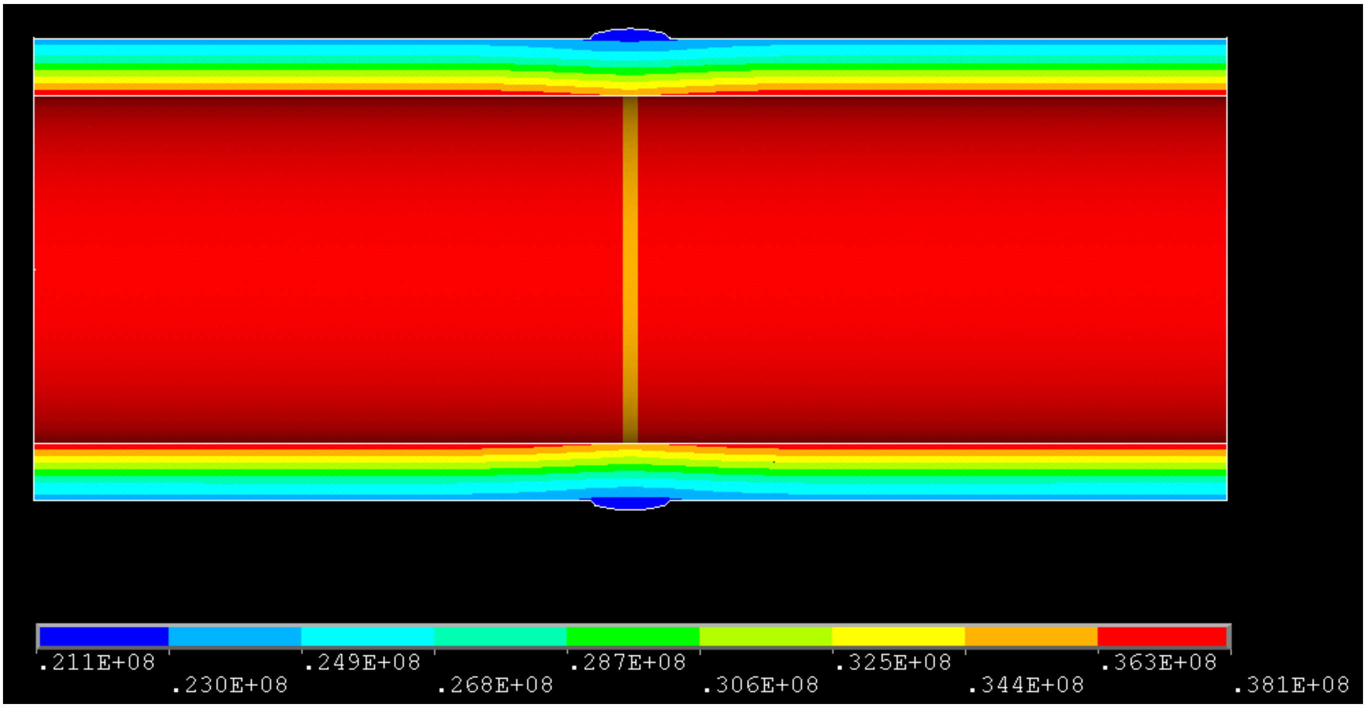

Figure 12: No Residual Stress Pipeline Weld Head Under 10Mpa Uniform Inner Pressure of Von Mises Stress Distribution

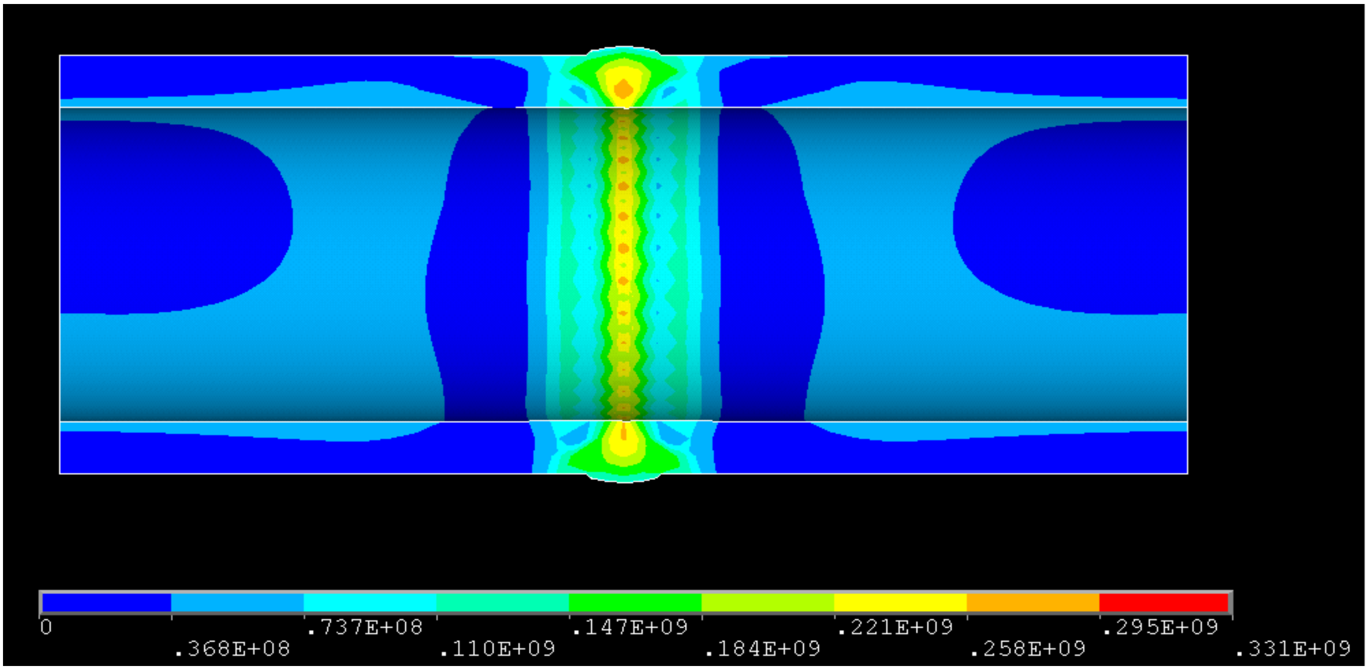

Figure 13: Pipeline Weld Head with Residual Stress Under 10Mpa Uniform Inner Pressure of Von Mises Stress Distribution

Figure 14(a-d) fully releases no inner pressure and within inner pressure (5MPa, 10MPa, 15MPa), maximum, and average Von mises stress changes of four surfaces of three layers along with the pipeline axis. 8 figures and 32 curves reflects the two common characteristics of the Von mises stress changes, shown as follows.

(1) after 0.018m changing tends all curves become parallel which means residual stress not affecting the Von mises stress distribution of pipeline with inner pressure. From Figures we can conclude that the effect region of residual stress is the 0.018m of girth weld head center and extended to more general is that the ratio of axis distance from the circumferential weld Center to in pipe diameter is 37.5% and during the Von mises can be affected by the residual stress obviously.

(2) Those curves has two intersect point in axis pipeline. Before the first intersection point, residual stress is positive direction of overlay role on Von mises with inner pressure. Between the two intersect point, residual stress is negative direction of overlay role on Von mises with inner pressure. After the second intersection point residual stress is no effect on Von mises with inner pressure and the no Von mises changing trend is similar with no residual stress situation loading inner pressure.

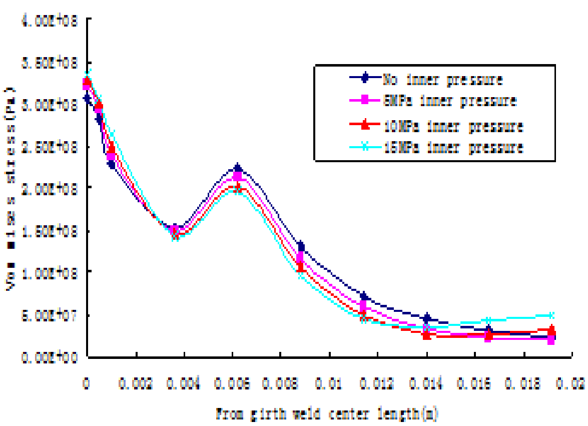

Figure 14A: The Maximum Von Mises Stress of Pipeline Weld Head Inner Surface from the Distance Along The Axis

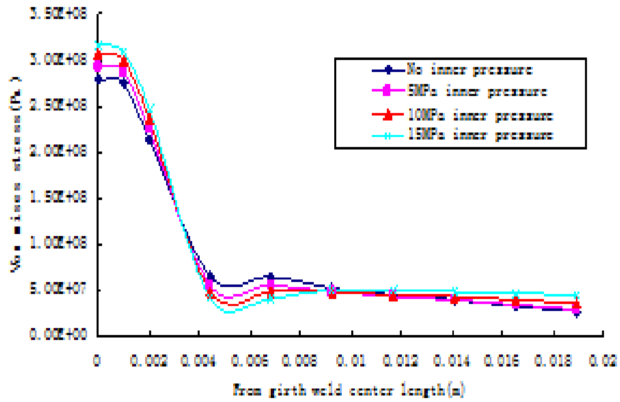

Figure 14B: The Maximum Von Mises Stress of Pipeline Weld Head First Layer Outer Surface from the Distance Along The Axis

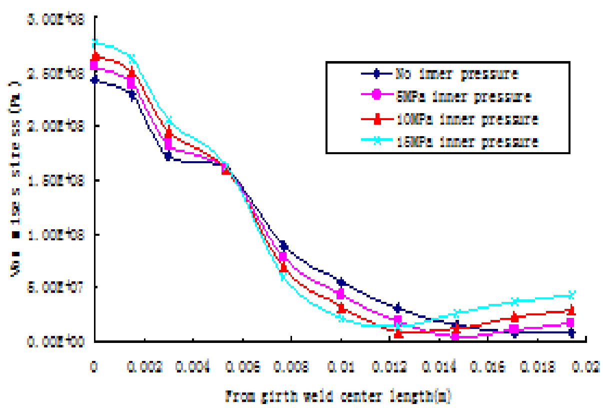

Figure 14C: The Maximum Von Mises Stress of Pipeline Weld Head Second Layer Outer Surface from the Distance Along The Axis

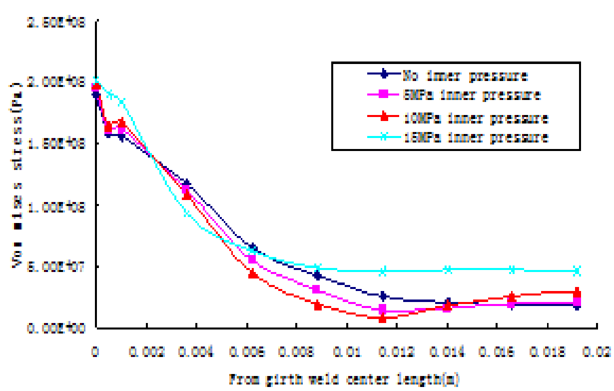

Figure 14d. The Maximum Von mises stress of Pipeline weld head third layer outer surface from the distance along the axis

Conclusion

This article compiles temperature field and stress field analysis program using APPL, and the arm of the two program is to achieve a finite element indirectly coupled numerical simulation of weld. A detailed analysis of the temperature field and residual stress field in pipe girth weld joints strength under different internal pressure can get the following several important points.

(1) By ANSYS "life and death" Element technology in pipeline girth welds, clearly temperature field changes in welds head can be Simulated. Because this simulation of pipeline girth weld layers is three, assume weld at layer *n*, and the thermal cycles number of the layer is 3-*n*+1, which is similar with Figures 7 and 8 in reference (Zhao, 2011).

(2) With simulating the pipeline effect area of residual stress under different inner pressure can be clearly observe, and the area is from the circumferential weld Center approximately 0.0018m.

References

Chen LW (2010) *Study on Numerical Simulation and Control of Plate Element Welding Deformation*. Wuhan Polytechnic University. Wuhan, china, (Unpublished Master Thesis).

Dar NU; Qureshi EM; and **Hammouda MMI** (2009) Analysis of Weld-Induced Residual Stress and Distortions in Thin-Walled Cylinders. *Journal of Mechanical Science and Technology*, **23** (4): 1118-1131.

Dong P (2007) On the Mechanics of Residual Stresses in Girth welds. *Journal of Pressure Vessel Technology*, **129** (3): 345-354.

Dong P (2001) Residual Stress Analyses of a Multi-Pass Girth Weld:3-D Special Shell Versus Axisymmetric Models. *Journal of Pressure Vessel Technology*, **123** (2): 207-213.

Dong P; Zhang J (2002) Effects of Repair Weld Length on Residual Stress Distribution. *Journal of Pressure Vessel Technology*, **124** (2): 74-80.

Feli S; Aaleagha MEA; Foroutan M; and **Farahani EB** (2012) Finite Element Simulation of Welding Sequences Effect on Residual Stresses in Multipass Butt-Welded Stainless Steel Pipes. *Journal of Pressure Vessel Technology*, **134** (1): 0112091.1-0112091.9 pages. doi:10.1115/1.4004571.

Feng XY (2006) *Based on ANSYS Numerical Simulation of Welding Temperature Field and Stress*. Wuhan Polytechnic University. Wuhan, china, (Unpublished Master Thesis).

Hossain S; and **Truman CE** (2006) Measurement of Residual Stresses in A Type 316H Stainless Steel Offset Repair in a Pipe Girth Weld. *Journal of Pressure Vessel Technology*, **128** (8): 420-426.

Jiang W; Yahiaoui K; and **Hall FR** (2005) Finite Element Predictions of Temperature Distributions In a Multipass Welded Pipint Branch Junction. *Journal of Pressure Vessel Technology*, **127** (1): 7-11.

Karlsson RI; and **Josefson BL** (1990) Three-Dimensional Finite Element Analysis of Temperatures and Stresses in A Single-Pass Butt-Welded Pipe. *Journal of Pressure Vessel Technology*, **112** (1): 76-84.

Liu Y; Yin; Arendt XP; Chen W; and **Huang HZ** (2010) A Hierarchical Statistical Sensitivity Analysis Method for Multilevel Systems with Shared Variables. *Journal of Mechanical Design*,, **132** (3): 031006.1-031006.11.

Ming L; and **Tang JY** (1998) Study on Welding Stress of Multi-Layer Welding Mild Steel Pipe. *Petroleum Engineering Construction*, **1** (1): 1-4.

Ming P (2010) The New Advance of Chinese Oil and Gas Pipeline in 2009s (in Chinese). *International Oil Economics*, **3** (1): 14-19.

Mochizuki M; Hayashi M; and **Hattori T** (2000) Residual Stress Distribution Depending on Welding Sequence in Multi Pass Welded Joints with X-Shaped Groove. *Journal of Pressure Vessel Technology*, **122** (1): 27-32.

Sddique M; Abid M; Juneio HF; and **Mufti RA** (2005) 3-D Finite Element Simulation of Welding Residual Stresses in Pipe-Flang Joints: Effect Of Welding Parmeters. *Materials Science Forum*, **490** (6): 79-84.

Wang HF; and **Sang ZF** (2011) The Effect of the Welding on the Residual Contact Stress in the Expanded Zone of Expanded Welded Tube to Tubesheet Joints. *Journal of Pressure Vessel Technology,* **133** (6): 0612092.1-0612092.6 pages. doi:10.1115/1.4005252.

Wang Z; Huang HZ; and **Liu Y** (2010) A Unified Framework for Integrated Optimization under Uncertainty. *Journal of Mechanical Design*, **132** (5): 051008.1-051008.8.

Xia BY (2007) T-joint Welding Numerical Simulation of Temperature Field and Stress Field. Zhejiang University, Hangzhou, china, (Unpublished Master Thesis).

Zhang X; Huang HZ; and **Xu H** (2010) Multidisciplinary Design Optimization with Discrete and Continuous Variables of Various Uncertainties. *Structural and Multidisciplinary Optimization*, **42** (4): 605-618.

Zhao HH; Zhang G; Yin Z; and **Wu L** (2011) A 3D Dynamic Analysis of Thermal Behavior during Single Pass Multi Layer Weld-Based Rapid Prototyping. *Journal of Materials Processing Technology*, **211** (3): 488-495.