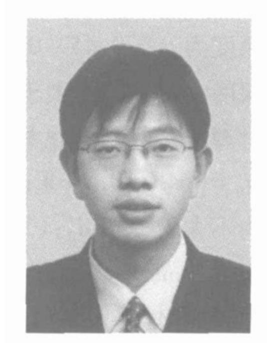
第 27卷 第 3期
2006年 3月
焊 接 学 报
TRANSACTIONS OF THE CHINA WELDING INSTITUTION
Vol 27 No. 3
March 2006

海底管线用钢低温断裂韧度的试验研究和数值模拟

柳锦铭， 陈 通
(天津大学 工业工程系，天津 300072)

摘 要：根据 BS7448断裂韧度试验标准，对 X65管线钢焊接接头的低温（ −15℃）裂纹尖端张开位移（CTOD）进行了测试并对其进行了三维有限元数值模拟。试验取标准三点弯曲试样，然后根据试验得到的 p−V曲线来计算 CTOD 值，并对试验结果进行了讨论和总结。尤其是对试验后试样的断面分析，验证了试验结果的有效性，从而解释了试验值分散性大的问题。通过有限元分析得到了计算的 p−V曲线，该 p−V曲线落在试验测得的 p−V曲线区间内，说明有限元分析的正确性。因此，在没有其它方法可获得的情况下，能够用有限元数值模拟进行分析计算。

关键词：裂纹尖端张开位移（CTOD）；断裂韧度；有限元；数值模拟
中图分类号：TG405 文献标识码：A 文章编号：0253−360X（2006）03−101−04


柳锦铭

0 序 言

X65管线钢作为海底油气管道的主要材料，承担着输送石油、天然气以及易燃、易爆介质等任务[1]。由于常常处在低温等恶劣的环境下服役，所以良好的低温韧性是保证管道安全运行的重要因素之一。由于在管道焊接过程中，常常使焊接接头的组织性能劣化及产生缺陷，使焊接缺陷处成为整个压力管道中最薄弱的部位，易产生裂纹启裂、扩展甚至失稳断裂[2]。因此有必要对 X65管线钢焊接接头的低温韧性进行研究。

CTOD 作为一种重要的弹—塑性分析参量，已经得到了广泛的应用。近年来，在国际上由英国焊接研究所提出的测试断裂韧度 K_{IC}、CTOD（δ）和 J_{IC} 的英国统一试验标准 BS 7448受到国际焊接学会的重视并予以推广应用。目前已被国际标准局（ISO）采纳，编号为 ISO/TC164/SC4—N400。并且该标准提供了判断结果有效性的方法，尤其是针对热影响区（HAZ）试件，对于选取材料和安全评定起着非常重要的作用。

应用 BS 7448标准来测试 X65管线钢焊接接头 CTOD 断裂韧度，并在试验的基础上应用有限元分析软件 ANSYS进行了数值模拟。比较试验和计算结果，从而验证数值模拟方法的可靠性，为断裂韧度的确定提供一种计算方法。

收稿日期：2005−04−13

1 CTOD 断裂韧度试验研究

根据英国标准 BS 7448，制备带预制疲劳裂纹的三点弯曲（TPB）标准试样。其中母材根据 BS 7448：1991−PartI "确定金属材料 K_{IC}、临界 CTOD 和 J积分的方法"[3]，焊缝和 HAZ根据英国标准 BS 7448：1997−Part II "确定金属材料 K_{IC}、临界 CTOD 和 J积分的方法"[4]。试验过程当中自动记录了 p−V曲线。试样形式见图 1。

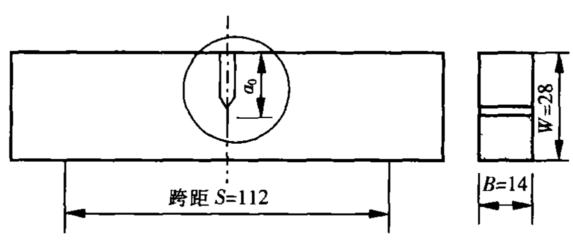


图 1 三点弯曲试样
Fig. 1 Three points bending specimen

1.1 试验过程

试验采用一次加载方式直到试样失稳破坏，加载速率控制在 0.5～1.0mm/min范围内，并同时记录试样载荷—位移曲线。

试样失稳破坏后，从低温箱中取出，对断口进行烘干处理后在试验机上快速压断试样。从断裂试样上取下断口用工具显微镜测量试件的裂纹长度 a_0，

具体方法为,沿试件厚度方向取 9个测试位置分别测量,其中最外侧的两个点位于距试件表面 1% B处,然后在这两个点之间等间距的取 7个测试位置。

根据 $p-V$ 曲线和相关材料性能数据计算 *CTOD* 断裂韧度值。

1.2 有效性判别

BS 7448标准对母材金属有效试样规定,平均裂纹深度为 $a_0=0.45\sim0.55W$;裂纹前缘任意两个裂纹深度的差值均不大于 10% a_0。

对焊接接头断裂试样规定:(1)试样平均裂纹深度 $a_0=0.45\sim0.70W$;(2)任意两个裂纹深度的差值均不大于 20% a_0;(3)预制疲劳裂纹的最小值不小于 1.3 mm 和 2.5% W 二者中的较大值;(4)预制疲劳裂纹时,其开裂方向平面与线切割裂纹垂直延伸平面夹角不大于 10°;(5)对于热影响区试件,还需对断口晶粒进行金相分析,从而确定裂纹尖端在粗晶区的试验结果才为有效。

1.3 试验结果

综合以上计算和分析,得到 *CTOD* 试验结果见表 1。

表 1 *CTOD* 试验结果(−15 ℃)

Table 1 *CTOD* test results(−15 ℃)

切口位置	试样编号	*CTOD* δ_m/mm	有效性判断
焊缝	W−1	0.336	有效
	W−2	0.324	有效
	W−3	0.348	有效
热影响区	H−1	0.489	有效
	H−2	0.398	有效
	H−3	0.721	无效
	H−4	0.562	有效
	H−5	0.703	无效
	H−6	0.721	无效
母材	B−1	0.723	有效
	B−2	0.706	有效
	B−3	0.695	有效

根据试验结果可以看出,在 −15 ℃下 X65管线钢焊接接头的断裂韧度比较高。与母材相比,焊缝的 *CTOD* 值偏低。HAZ的 *CTOD* 值有很大的分散性,经过断面分析发现,当裂纹尖端位于粗晶区是 *CTOD* 值偏低;反之位于细晶区时偏高。因此,试验后的断面分析,确定裂纹尖端位置是判断结果有效性的一项重要措施。

2 有限元数值模拟分析

在试验基础上,利用大型 ANSYS有限元分析软件,采用三维有限元数值方法,对三点弯曲试样的应力场进行分析。

有限元法计算断裂参数的步骤为:(1)对含裂纹体划分单元;(2)指定约束及载荷;(3)进行弹性、弹塑性分析求出各节点的位移、应力和应变;(4)最后根据上述方法得到 *COD* 值。

在对裂纹体划分单元时,首先遇到的问题是裂纹尖端应力、应变场的奇异性。当用普通单元构造裂纹尖端区域时,要求计算单元多,在尖端附近单元划分必须很密(单元尺寸为 $10^{-2}a$, a 为裂纹长度),才能有足够的精度。目前计算线弹性断裂力学参量 K 和弹塑性断裂力学参量 *COD*、J 等时,广泛采用的是退化等参奇异单元法或虚拟裂纹扩展法[5~7]。

ANSYS有限元软件提供了用于空间分析的退化 20节点等参奇异单元 SOLID 95,见图 2。它是由普通 20节点等参元退化而成的,将其中一个面退化为一条线,将该线设为裂纹的前沿,再将临近该线的四个中间节点移到 1/4边长处,就构成了退化 20节点奇异等参元。通过这种构造而成的单元,能够满足裂纹尖端的奇异性,可适用于线弹性及弹塑性断裂力学参数的计算。

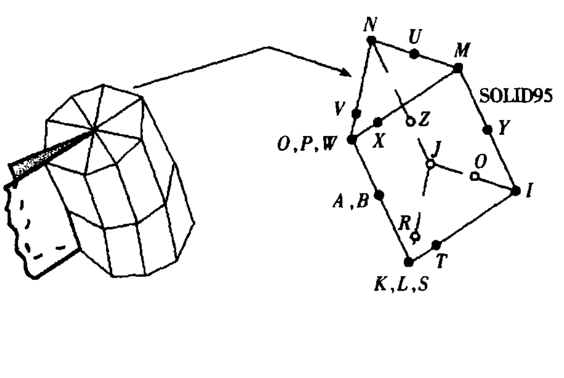


图 2 裂纹尖端单元 SOLID95

Fig. 2 Element SOLID95 at tip of crack

对含裂纹体划分单元时,围绕裂纹尖端的第一排单元采用上述三维退化等参奇异单元 SOLID 95,和它相连的外层单元仍采用普通 8节点实体单元 SOLID 45。奇异单元的优点是裂纹尖端处单元划分不必太细,就能满足计算精度的要求。

2.1 三点弯曲试样有限元模型的建立

用于有限元计算的三点弯曲试件由于对称性,取试样的 1/4建模。焊缝和母材都选用双线性随动

强化的应力应变关系，其弹性模量为 2.1×10^5 MPa，泊松比为 0.3，计算时采用 Von Mises屈服准则及相关流动准则。

裂纹尖端的最小尺寸为 0.013 mm × 0.069 mm × 0.5 mm，其局部网格划分如图 3所示。沿试件厚度方向从试样表面到试样中部共分四层，分别为 0.5 mm，1.0 mm，2.0 mm，3.5 mm，共分 6 965 个单元，8 700个节点，总体网格见图 3。采用不均匀划分主要是由于表面拘束小，应力梯度大，因此表层处网格密，而试样中部，应力梯度小，因此试样厚度中心网格疏。由于断裂控制区集中于裂纹尖端附近，所以裂纹尖端网格较密，其余地方较疏，这样可以减少单元，从而节省计算时间。

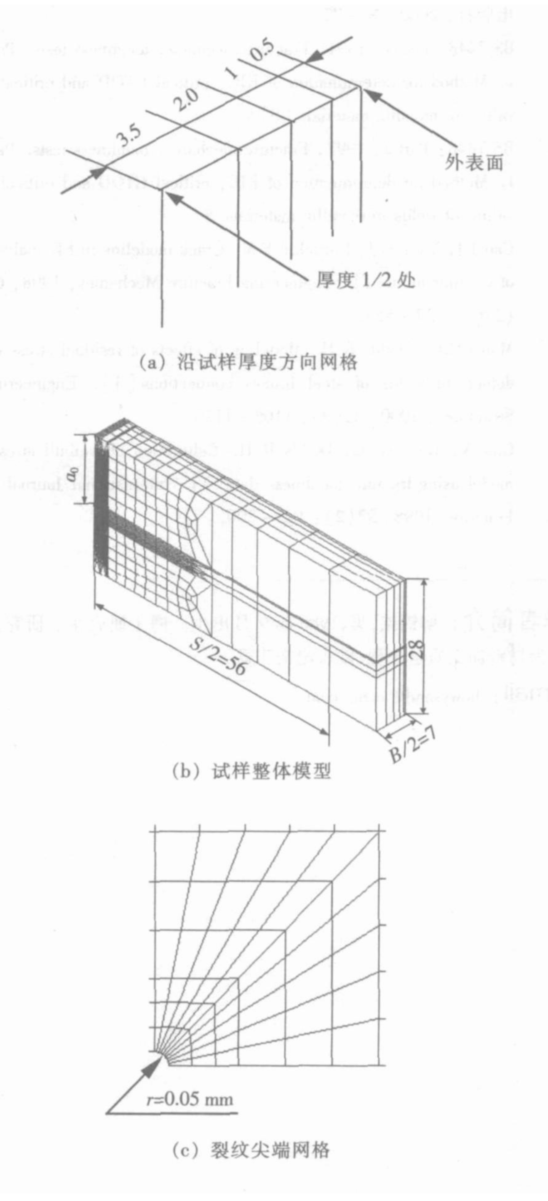


图 3　有限元模型整体及局部网格

Fig. 3　Global and local modes

2.2　三点弯曲试样裂纹尖端应力场分布

采用裂纹尖端张开位移 *CTOD* 表征变形水平，研究在不同变形水平下，沿不同方向应力分布。*CTOD* 的计算方法和试验中采用的计算方法相同，即根据计算结果整理出载荷—位移 ($p-V$)曲线，根据公式计算每一载荷步所对应的 *CTOD* 值。从有限元计算出的 $p-V$曲线和试验中测得的 $p-V$曲线图(图 4)可以看出，计算出的曲线落在试验曲线区间内，说明有限元分析的正确性。

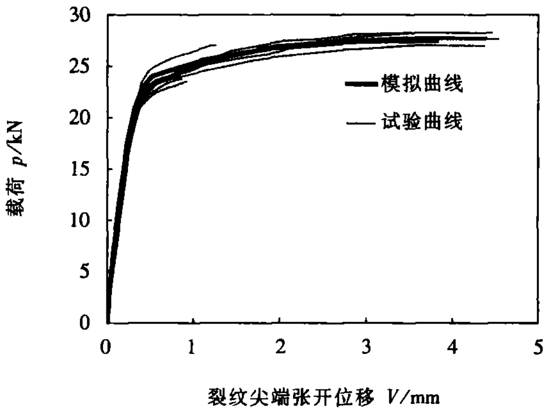


图 4　模拟和试验的 $p-V$ 曲线

Fig. 4　Simulated and experimental $p-V$ curves

2.3　裂纹尖端沿裂纹长度方向应力分布

图 5给出了在不同变形水平下(即裂纹尖端张开位移分别为 $\delta=0.106$ mm和 0.358 mm)三点弯曲试样板厚中部裂纹尖端附近最大主应力，图中横坐标为裂纹长度方向上距裂纹尖端的距离。

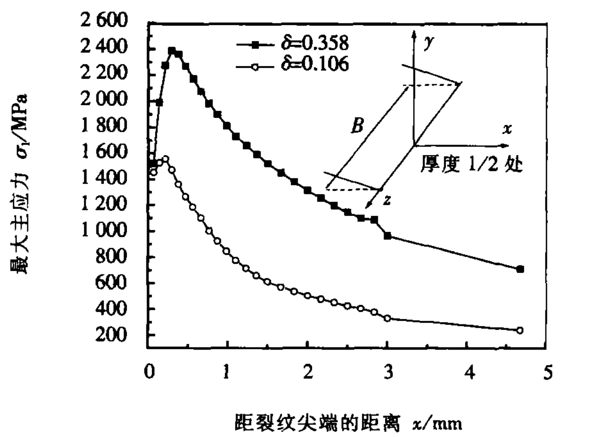


图 5　沿裂纹方向最大主应力分布

Fig. 5　Maximum principal stress distribution along crack

由图可见，同一变形水平下裂纹尖端最大主应力沿裂纹方向先增大后减小，距尖端较远处趋于平

稳。在同一位置,变形水平越高,最大主应力越大,则发生断裂的可能性越大,由此定性说明了变形越大越易断裂这一现象。

2.4 裂纹尖端沿厚度方向应力分布

图 6给出了在如前所述的载荷水平下,三点弯曲试样裂纹尖端附近最大主应力沿试件厚度方向的分布。

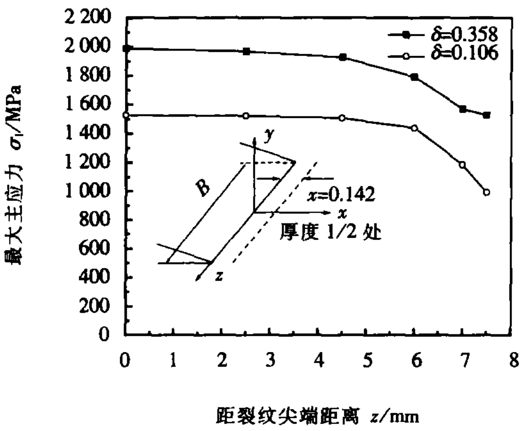


图 6 沿厚度方向最大主应力分布

Fig. 6 Maximum principal stress distribution along thickness

由图可见,同一载荷水平下裂纹尖端最大主应力从中心沿厚度方向逐渐减小,载荷水平越高,最大主应力越大。

3 结 论

(1) 试验得到了 X65管线钢焊接接头低温(-15 ℃)*CTOD* 断裂韧度值。*CTOD* 值由高到低的顺序为母材>HAZ>焊缝。

(2) 对于热影响区 *CTOD* 试验,试样的断口分析判断裂纹尖端的位置是验证结果有效性的关键。结果与 $p-V$曲线和 *CTOD* 值相一致,解释了试验值分散性大的问题。

(3) 从有限元模拟结果可以看出,计算出的 $p-V$曲线落在试验测得的 $p-V$曲线区间内,说明有限元分析的正确性。因此,在没有其它方法可获得的情况下,能够用有限元数值模拟进行分析计算。

参考文献:

[1] 马 良. 海洋油气管道工程[M]. 北京: 海洋出版社, 1987. 1-30.

[2] 霍立兴. 焊接结构的断裂行为及评定[M]. 北京: 机械工业出版社, 2000. 46-77.

[3] BS 7448. Part 1: 1991, Fracture mechanics toughness tests. Part 1. Method for determination of KIC, critical CTOD and critical J values of metallic materials[S].

[4] BS 7448. Part 2: 1997, Fracture mechanics toughness tests. Part 1. Method for determination of KIC, critical CTOD and critical J values of welds in metallic materials[S].

[5] Cao J J, Yang G J, Burdekin F M. Crack modeling in FE analysis of circular joints[J]. Engineering Fracture Mechanics, 1998, 61(5/6): 537-553.

[6] Matos C G, Dodds R H. Modeling of effects of residual stress on defects in welds of steel frames connections[J]. Engineering Structures, 2000, 22(9): 1103-1120.

[7] Gao X, Ruggieri C, Dodds R H. Calibration of weibull stress model using fracture toughness data[J]. International Journal of Fracture, 1998, 92(2): 175-200.

作者简介: 柳锦铭, 男, 1980年 9月出生。博士研究生。研究方向为材料加工质量工程, 发表论文 3篇。

Email: howysand@sohu.com

The fracture parameter crack opening displacement and its compound angles were also calculated. The simulated results show that the effects of load angles on the welded joints are more significant than that of strength matching parameters when the loading angle is small. But the effects of strength matching on welded joint are more evident than that of the loading when the loading angle exceeds 30 degree.

Key words: finite element method; welded joint; compound angle; loading angle

Effect of specimen dimension on welding residual stresses ZHOU Jian-xin[1], XU Hong[1], WANG Jun-sheng[2], LI Dong-cai[3], ZHANG Li[1], LIU A-long[1] (1. East China University of Science and Technology, Shanghai 200237, China; 2. Liaohe Petroleum Exploration Bureau, Panjin 124010, Liaoning, China; 3. Xi'an Petroleum University, Xi'an 710065, China). p96－100

Abstract: The magnitude and the distribution of welding residual stress of butt thin plates were calculated by means of the large scale finite element method software ANSYS, and the effect of plate length and plate width on welding residual stress was obtained, especially the effect of the middle sized plate and narrow plate on welding residual stress. The results show that when the weld length is less than 100mm, the longitudinal residual stress of different width plate is the same, and all kinds of plates belong to wide plate. With the increment of the weld length, the distribution of the residual stress gradually transfer to middle-sized plate and narrow plate. The calculated resuls was compared with the experimental results, and the results were identical. The acquired results may be used to predict the welding residual stresses.

Key words: welding; residual stress; finite element method; effect of specimen dimension

Numerical simulation and study on fracture toughness of submarine pipeline LIU Jin-ming, CHEN Tong (Department of Industrial Engineering, Tianjin University, Tianjin 300072, China). p101－104

Abstract: In accordance with the BS7448, *CTOD* (crack tip opening displacement) tests were conducted at －15℃ for welded joints of X65 pipeline steel. The three-dimensional finite element numerical simulation was performed on the *CTOD* specimen. According to the requirements of the standard, the specimen was a standard three point bending shape. Then the *CTOD* test were performed, and the results were calculated using the *p-V* curves. Finally, The test results were generalized and analysed. Especially, the observation of a slice containing the fracture face explained the dispersity of the test values, and validated the test results. From the results of the finite element analysis, the simulated *p-V* curve was obtained. It could be seen that the simulated curve was within the experimental curves, so the numerical simulation was accurate. All the above indicate that it is a practical method to calculate *CTOD* fracture toughness by finite element numerical simulation when no other methods are available.

Key words: crack tip opening displacement; fracture toughness; finite element; numerical simulation

Modeling and simulating of temperature field of laser welding for stainless steel HAN Guo-ming, LI Jian-qiang, YAN Qing-liang (School of Materials Science and Engineering, Tianjin University, Tianjin 300072, China). p105－108

Abstract: Temperature field of laser welding stainless steel 304 was dynamically simulated with finite element analysis software－ANSYS. Effects of temperature correlation of the physical parameters of material, latent hear of fusion, convection and radiation on the temperature field of laser welding were analyzed. In view of the characters of laser welding, a heat source combined the body loads were designed. Considering the high non-linear of the solution and the small focal diameter, a serial of measures were adopted, such as modeling with pyramid, stepped loading and solution. In the load history, an overmeasure control method was taken to ensure the precision of nodal selection. It was shown that the simulation results of weld shape were in accordance with the experimental results, the isotherms looked like ellipse and were dense at the head of the moving heat source. And the temperature gradient was high at the head of the heat source. But at the end of the moving heat source, the isotherms were sparse and the temperature gradient was low. Temperature near the surface of the weld pool was the highest, and up to about 13 332 ℃.

Key words: stainless steel; laser welding; temperature field; numerical simulation

Mechanisms of rotational magnetic field stirring of laser welded non-magnetic alloy for laser welding YU Sheng-fu[1], ZHANG You-shou[2], LEI Yi[3], XIE Zhi-qiang[2], WU Dong-zhou[2], LIU Ping[2] (1. Huazhong University of Science and Technology, Wuhan 430074; 2. China Academy of Engineering Physics, Mianyang 621900, Sichuan, China; 3. Huadong Petroleum University, Dongying 257061, Shandong, China). p109－112

Abstract: The influences of rotational magnetic field on microstructures of Al-12Si alloy for laser welding have been analyzed. The test results show that the rotational magnetic field can change the behaviors in motion of liquid Al-12Si alloy, refine the weld metal grain and eliminate the drawbacks in Al-12Si weld metal, although laser beam has nonmagnetism and is not conductor, and Al-12Si alloy is nonmagnetic material. For laser welding, induction current of liquid metal in welding pool is caused, and the electromagnetic stirring force on liquid metal with electric charge lead to the rotational motion of liquid metal in welding pool when the rotational magnetic field passes liquid metal. The higher the rate of the rotational magnetic field is, the larger the electromagnetic stirring force on liquid metal is.

Key words: magnetic stirring; rotational magnetic field; laser welding; Al-12Si alloy