International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 12, December-2014

Stress, Strain and Displacement Analysis of Individual Parts and the Pipe of an Autonomously Designed Pipe Bending Machine using Finite Element Analysis

Mohammed Noorul Hussain
Department of Mechanical Engineering, B.E
Muffakham Jah College of Engineering and Technology
Hyderabad, India 500094

Syed Ali Sufiyan
Department of Mechanical Engineering, B.E
Muffakham Jah College of Engineering and Technology
Hyderabad, India 500094

Mohammed Nizamuddin Ahmed
Department of Mechanical Engineering, B.E
Muffakham Jah College of Engineering and Technology
Hyderabad, India 500094

Kosuri Harish Varma
Department of Mechanical Engineering, B.E
Muffakham Jah College of Engineering and Technology
Hyderabad, India 500094

***Abstract*----In our everyday lives, bent tubes are required for the transportation of all kinds of fluids or gases and also used as construction elements in almost any industrial branch. They are just as wide spread in the car, aviation and shipbuilding industry as they are in the chemical industry, refrigeration and air-conditioning technology, furniture industry, steel constructions or machine technology and plant engineering. In pipeline construction or in the installation business, where bending as a production method has been neglected for a long time and conventional- often expensive- connection techniques (e.g. welding) were used instead. Bending has gained importance in the past years, due to its large variety of advantages. This is because the modern tube bending machines have made it possible to optimally combine processing precision with productivity, not forgetting the high operation comfort. Considering the importance of pipe and with a will to contribute to efficient engineering in the world we have designed a pipe bending machine which can be used to bend solid and hollow pipes of 20mm – 40mm Dia. The purpose of this work is to analyze the stress occurring in its parts to make the design sturdy enough and to investigate the stress distribution in the pipe to reduce the defects in order to make the machine more efficient.**

Keywords: Compression, bending, thickening, ovality, thinning, bending, stress

1. INTRODUCTION

Pipe bender has three main components namely the primary roller, the bending roller and the assembly of roller and handle extension. When the process of bending is carried out huge amount of force is applied on these parts hence great stresses are developed within these parts[2]. The material selected was low tensile mild steel (1020 Carbon Steel) with yield strength of 620 Mpa. Stress analysis of these parts is important to identify the areas of stress concentration to prevent failure of these parts and increase their life. The models of these parts were created using **'Solidworks 2011'** software and the stress analysis was carried out using **'Solidworks Simulation'** feature which performs **Finite Element Analysis** on the components to determine nodal stresses.

2. ANALYSIS OF PRIMARY ROLLER

Roller is the stationary part in the pipe bender. It is a part on which maximum forces are applied. It has to withhold bending, and compressive stresses. This is the primary part of the bender which is an assembly of the handle extension and the main roller (through TIG welding) hence the joint also should be able to withhold the forces. The diameter of this roller was selected as 140mm, which is the direct bend radius of the pipe. Analysis of this part was done after considering two cases

CASE-1) Bending force of 5000N on the part (after fixing) & joint strength testing

CASE-2) Compressive force 5000N on roller face due to pipe bending force application.

CASE-1) Bending test

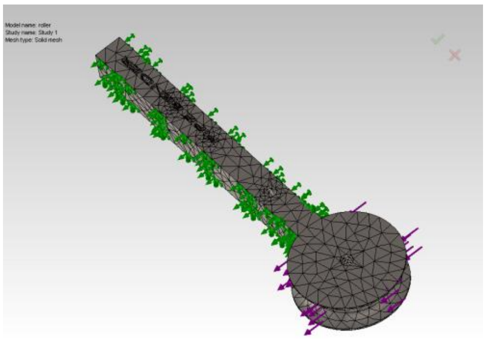


Figure 2.1: Meshed Model

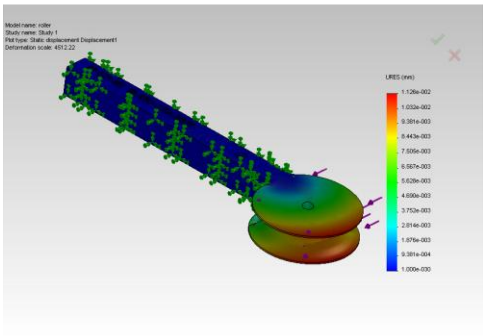


Figure 2.2 Distribution spectrum

CASE-2) Compressive test

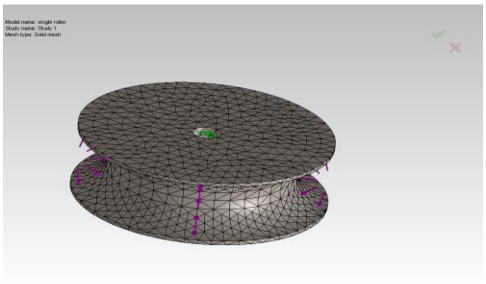


Figure 2.3 Meshed Model

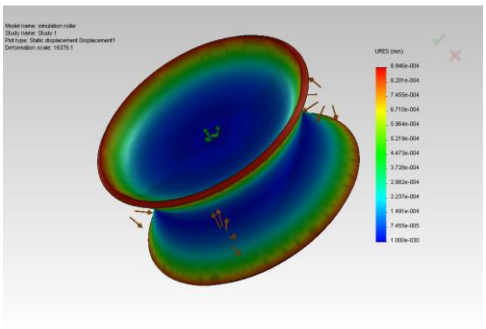


Figure 2.4 Displacement Spectrum

3. ANALYSIS OF BENDING ROLLER

Bending roller is the component through which effort applied for bending is passed through the pipe. It also guides the pipe through a circular path to get the bend correctly. The internal circular groove in this roller helps to hold the pipe while bending and balance the stresses occurring in the pipe to help retain the pipe's circular cross-section and reduce fracture and other defects in the pipe. The diameter of this roller was selected as 60mm. The roller is analyzed for compressive stresses which act upon the internal circular face which is in contact with the pipe while bending.

CASE-1) Compressive test

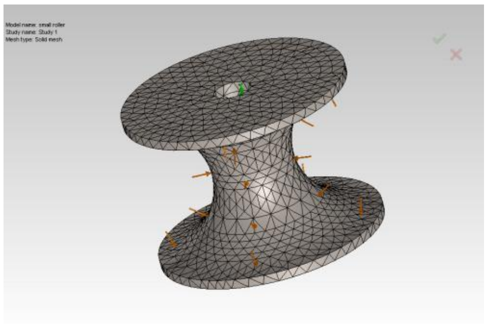


Figure 3.1 Meshed Model

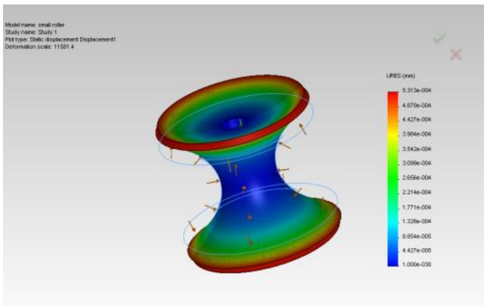


Figure 3.2 Distribution Spectrum

CASE-2) Bending test

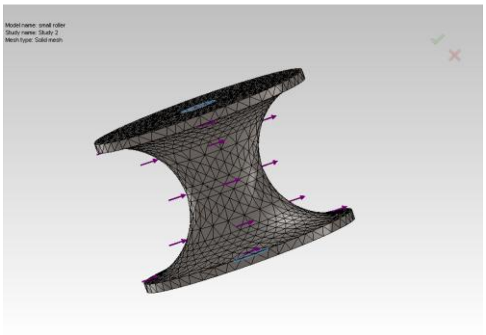


Figure 3.3 Meshed Model

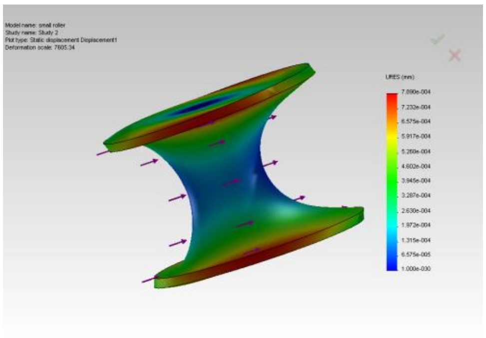


Figure 3.4 Distribution Spectrum

4. ANALYSIS REPORT OF ROLLERS

YIELD POINT STRESS= 620 MPa [1]		PRIMARY ROLLER		BENDING ROLLER	
		CASE-1 LOAD= 5000N	CASE-2 LOAD= 5000N	CASE-1 LOAD= 5000N	CASE-2 LOAD= 5000N
DISPLACEMENT (mm)	MAX.	**1.12 x 10^{-2}**	**8.9 x 10^{-4}**	**5.3 x 10^{-4}**	**7.8 x 10^{-4}**
	MIN.	**1.03 x 10^{-3}**	**7.4 x 10^{-5}**	**1 x 10^{-3}**	**1 x 10^{-5}**
STRESS (MPa)	MAX.	**24.134**	**2.864**	**2.085**	**10.525**
	MIN.	**0.00**	**0.00**	**0.00**	**0.00**
STRAIN	MAX.	**9.86 x 10^{-5}**	**9.5 x 10^{-6}**	**7.4 x 10^{-6}**	**3.5 x 10^{-5}**
	MIN.	**0.00**	**0.00**	**3.7 x 10^{-7}**	**8.7 x 10^{-7}**

Table 4.1 Results.

The results of the finite element analysis as tabulated in table 3.1 show that in no case in either the primary or the beding rollers the stress exceeds that yield stress value of the material of the rollers. Considering this analysis it is safe to say that the design of the rollers is sustainable and safe for use of operation and fabrication.

5. STRESS ANALYSIS ON PIPE

All existing works with regard to the determination of plastic loads, assume the cross-section of the pipe bend to be circular with uniform thickness. In reality, the pipe bend exists with shape imperfections namely ovality and thinning/thickening as the result of the bending process. The acceptability of pipe bend depends on the magnitude of ovality and thinning. Therefore, it is more relevant to perform stress analysis on the pipe to pre-determine the extent of ovality in cross-section of the pipe bend.

Prior to stress analysis it is important to know a few definitions and formulas.

Ovality: The degree of ovality is determined by the difference between the major and minor diameters divided by the nominal diameter of the pipe [3]. When expressed in percentage form

$$C_o = \frac{D\,max - D\,\min}{D}\; x\;100$$

where, $D = \dfrac{D\,max + D\,min}{2}$

Thickening: Thickening occurs at intrados and is defined as the difference between the maximum thickness and the nominal thickness divided by the nominal thickness of the pipe bend. The percentage thickening is given as

$$C_{th} = \frac{t\,max - t}{t}\,x100$$

Thinning: Thinning, which occurs at extrados of the pipe bend, is defined as the ratio of the difference between the nominal thickness and the minimum thickness to the nominal thickness of the pipe bend and is expressed in percentage

$$C_t = \frac{t - t\,min}{t}\; x\;100$$

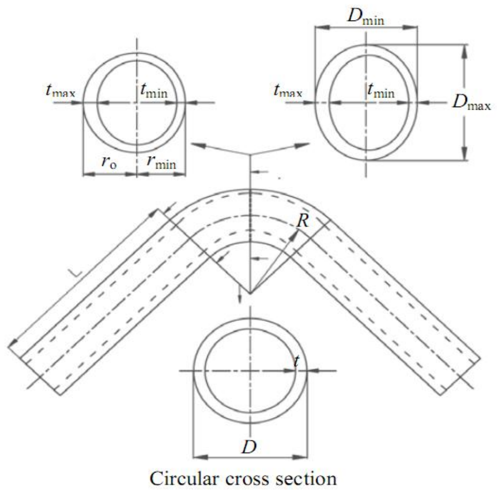


Figure 5.1

The first and foremost step in stress analysis of the pipe is the determination of the load which produces a considerable amount of deflection in the pipe when applied in perpendicular direction to the length of pipe. It is the same load that a human applies while bending the pipe. In pipe bending process the bend in pipe occurs due to the action of two different forces which act simultaneously on different sides of the pipe, They are namely the

1) COMPRESSIVE FORCE - This acts on the inner side of the pipe.[2]

2) TENSILE FORCE - This acts on the outer side of the pipe.[2]

To determine this, a pipe of ‘1 Inch’ diameter and thickness of 2 mm was modeled in solidworks. To this pipe, initially a compressive load of 100 N was applied on one side whereas a tensile load of 100 N was applied on the other. This condition did not seem to produce a considerable amount of deflection in the pipe, so the same step was repeated with different loads and eventually at a load of 2000 N results were considerably better. Acknowledging the fact that for acquiring a complete 90^0 bend more than 2000 N load is required, a load of 5000 N was considered apt for stress analysis.

The Centre part was constrained in all D.O.F to provide the restraint that occurs in actual bending process.

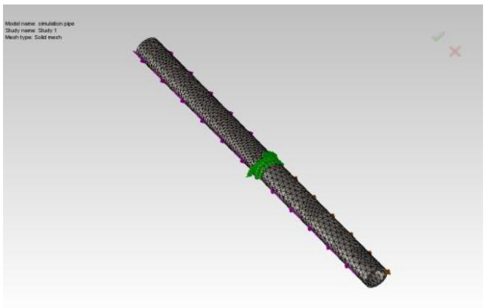


Figure 5.2 Meshed Model

The deflection plot is given below:

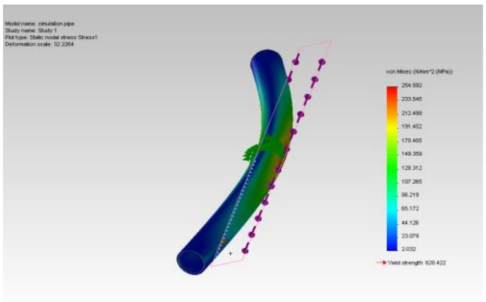


Figure 5.3 Deflection Spectruml

Stress analysis was done by virtually simulating the actual pipe bending process in ‘Solidworks Simulation’, a general finite element package. A no penetration contact set was given between the pipe and the primary roller. The roller was constrained in all degrees of freedom. No constraints were given to the pipe as in actual practice the pipe is not constrained. Forces were applied to one end of the pipe. Later a thickness analysis was performed using ‘Solidworks Design Evaluation’ to determine changes in thickness across the pipe bend. The material model was assumed to be elastic-perfectly plastic, and non-hardening J_2 flow theory was used. The material used was alloy steel with Young’s modulus (E), yield stress (σ_0) and Poisson’s ratio (υ) respectively as 210 GPa, 620 MPa and 0.28. The C3D20R, 20-node quadratic brick, reduced integration element, having three degrees of freedom (translation in x, y and z directions), was preferred in order to reduce computing time. Mapped meshing was used to generate the mesh model. The number of elements and nodes for each model were chosen as 13215 and 22586 respectively, after performing mesh refinements.

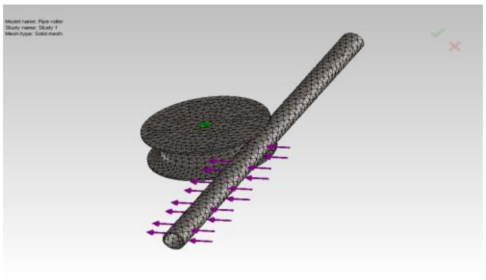


Figure 5.4 Meshed Model

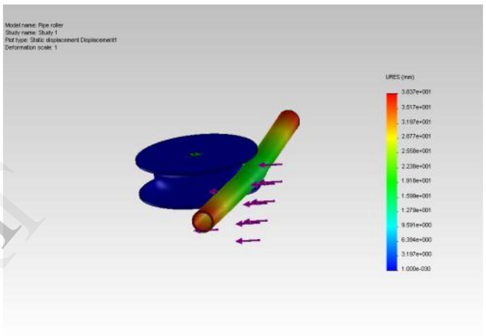


Figure 5.5 Distribution Spectrum

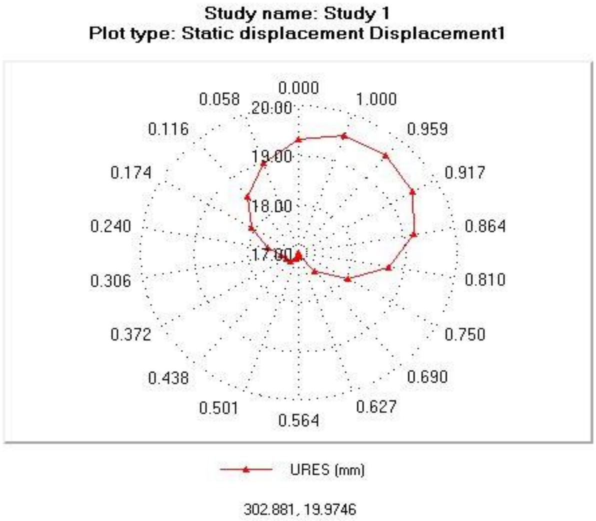
In the figure showing displacement, it is clear that the maximum displacement occurs at the ends of the pipe where the force is being applied. Here a maximum displacement of 40 mm was seen on application of 2000N force along the circular face of the pipe of 25.4 mm outer diameter and thickness of 2 mm. Due to the circularity of the primary roller the pipe acquires the contour of the roller on further application of force. To identify the extent of ovality in the pipe it is necessary to calculate displacements in the region where the pipe and the roller are in contact. This point acts as a fulcrum to bend the pipe but the reaction force that the roller applies acts opposite to the bending force and tends to cripples the cross section of the pipe. To examine this, a radar plot of displacement at the main edge of contact for this case was generated using ‘Solidworks Probe’ feature. In this radar plot a total of 16 nodes were considered along the circular edge of contact.

From the radar plot-1, it can be derived that the maximum displacement occurs in the outermost node and is in the range of 0.917 – 0.959 mm. This amount of displacement when compared to the bending displacement of the pipe is very less hence it can be said that in this setup the circular cross-section can be retained to a great extent.

Considering the stress distribution results of the analysis showed that maximum stresses occurred at the area of contact of the roller and pipe. In pipe bending the extrados of the pipe undergoes tension whereas the intrados of the

ISSN: 2278-0181
Vol. 3 Issue 12, December-2014

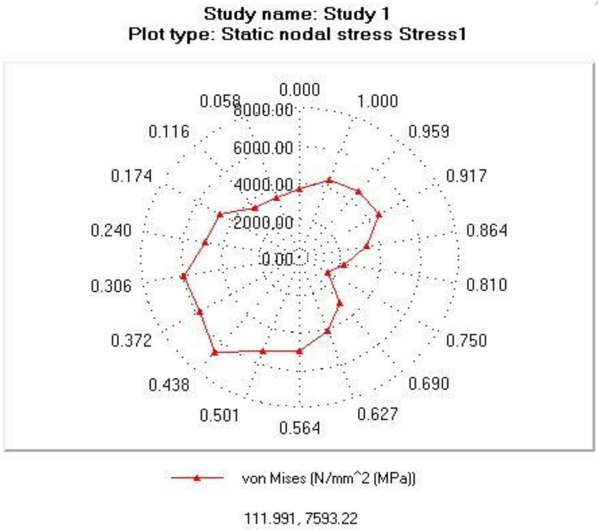
pipe undergo compression. Earlier results have showed that the tensile stress on the outer side of the pipe may cause



RADAR PLOT - 1

crippling and plastic failure of the pipe. Hence it is important to reduce these stresses. In actual practice the bending roller is provided to support the outer region of the pipe. The bending roller helps in reducing the stresses occurring and also guides the pipe along the circular contour. The bending roller itself has to bear compressive stresses whose analysis has been done earlier.

A radar plot of the stresses occurring along the contact area of pipe and area was generated with 18 nodes, which is as follows.



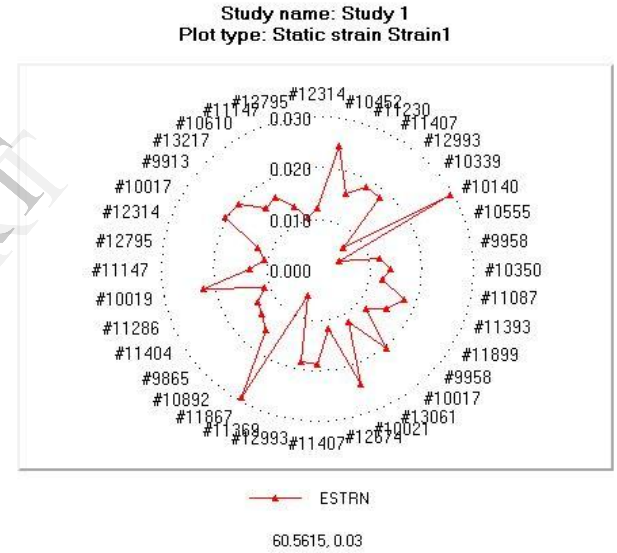
RADAR PLOT - 2

In the radar plot-2 we can see that the stresses in the outermost nodes have exceeded that yield point stress. But this is a case where no bending roller has been considered. Bending rollers are provided to absorb the tensile stress of the pipe and undergo compression. Theoretically bending rollers absorb 75 – 90 % of the reaction force of the roller and reduce a great amount of stress occurring in the pipe.

Considering the strain pattern of the pipe bending analysis showed that maximum strain occurred in the inner region of the pipe, at the region of contact and the exterior fibers of the pipe's exterior circumference.

Upon giving a load of 2000 N perpendicular the length of the pipe a maximum strain of 0.075 occurred in the inner region of the pipe i.e. at the area of contact of the pipe. And a minimum strain of 0.0063 occurred in the pipe.

A radar plot of stresses occurring in the area where stresses occurring were maximum was made by manually selecting 35 nodes was made, which is as follows



RADAR PLOT - 3

From the radar plot-3 it is clear that apart from a few peaks and lows the strain distribution is more or less uniform. This is essential as it helps in maintaining uniform cross-section of the pipe after bending.

6. RESULTS

The validity of the finite element analysis can be best proved by physical testing of full scale fabricated model. The model was fabricated at the machine tools lab under the mechanical department of Muffakham Jah college of Engineering. The machine was then tested using a successfully and several observations were made. The average values of the observations was calculated and used for calculation of crucial parameters that decide the credibility of the machine. The calculations showed that ovality percentage was under the safe value of 10% specified by the earlier conducted research[3].

6.1 CALCULATION OF OVALITY

The prerequisites for calculation of ovality are
1) Maximum Diameter - D_{max}
2) Minimum Diameter.- D_{min}
3) Average Diameter- D

From the Fig. We have D_{min} = 23.5 mm
D_{max} = 25.5 mm

$$D = \frac{D\,max + D\,min}{2} = \frac{23.5+25.5}{2} = 24.5mm$$

$$C_o = \frac{D\,max - D\,min}{D}\,x\,100 = \frac{25.5-23.5}{24.5}x100 = 8.16\ \%\ (oval)$$

6.2 THICKENING

The prerequisites for calculation of thickening are
1) Maximum Thickness- t_{max}
2) Minimum Thickness- t_{min}
3) Average Thickness- t

From the Fig. We have t_{max} = 2.99 mm
t_{min} = 1.01 mm

$$t = \frac{tmax+tmin}{2} = \frac{2.99+1.01}{2} = 2\ mm$$

$$C_{th} = \frac{t\,max - t}{t}\,x\,100 = \frac{2.99-2}{2}\,x\,100 = 49.1\ \%$$

6.3 THINNING

The prerequisites for calculation of thinning are
1) Maximum Thickness- t_{max}
2) Minimum Thickness- t_{min}
3) Average Thickness- t

From calculation of thinning we have t = 2mm
t_{min} = 1.01 mm

$$C_t = \frac{t - t\,min}{t}\,x\,100 = \frac{2-1.01}{2}\,x\,100 = 49.1\ \%$$

PARAMETER	PERCENTAGE
OVALITY (C_O)	8.16 %
THICKENING (C_{th})	49.1%
THINNING (C_t)	49.1%

REFERENCES

1. Miller, G. (2008). Justifying, Selecting and Implementing Tube Bending Methods. Metal Journal. Vol.1. Tubular Solution Inc. Ohio.
2. Khurmi, R.S. and Gupta, J.K. (2006). Workshop Technology (Manufacturing Processes), 6th Edition. Publication Division of Nirja Construction and Development Co. Ltd., New Delhi.
3. A R Veerappan and S Shanmugham on " Analysis for flexibility in the ovality and thinning limits of pipe bends" ARPN Journal of Engineering and Applied Sciences vol 3 no 1 February 2008.
4. Mohindar L. Nayyar. 2000. Piping Handbook, 7th Ed., McGraw-Hill. p A 269.
5. Lanhui Guo, Shijun Yang, Hui Jiao‖ Behavior of thin- walled circular hollow section tubes subjected to bending‖2013 ELSEVIER,PP 1-9 Shigley
6. J. E. "Mechanical engineering design." McGraw-Hill International Edition (1986). S. Baudin, P. Ray, B.J. Mac Donald, M.S.J. Hashmib, Development of a novel method of tube bending using finite element simulation, Journal of Materials ProcessingTechnology 153–154 (2004) 128–133.