Simulation of Pipe Bending Process with Abaqus

Satoshi ISHIKAWA[1], Yoshihiro ISHIKAWA[2]

[1]IDAJ Co., Ltd. and [2]Sango Co., Ltd.

***Abstract**: Exhaust manifolds are generally simple cast iron or stainless steel units which collect engine exhaust from multiple cylinders and deliver it to the exhaust pipe. The exhaust manifolds of steel pipe are formed with multiple bending operations. The pipe bending process is extremely complicated and the forming conditions such as controls of pressure are significant.*

This paper presents several Finite elements modeling techniques for simulating and optimizing pipe bending process using Abaqus/Explicit. Designing and tuning pipe-metal forming tools are quite complicated and time consuming tasks. These tasks must take into consideration a number of potential issues, such as the rate of forming, pipe shape smoothness, reduction ratio of metal sheet thickness, and pipe metal wrinkling. To shorten the design cycle and reduce the number of forming tool prototypes for the multi-process bending pipe, simulations with Abaqus served as virtual test grounds to provide valuable insight into the pipe's complex forming processes. Because of large deformation and contact interactions, Abaqus nonlinear capabilities were well suited for these tasks. This paper demonstrates Abaqus forming applications that helped resolve matters arising from realistic industrial forming design and production processes. Furthermore this paper argues the difference of three element types which are shell, continuum shell and solid. The analysis results were compared with the experimental thickness and the performance and accuracy were examined.

***Keywords**: Plastic forming, Element formulation, Friction, Explicit.*

1. Introduction

The exhaust manifolds of steel pipe are formed with multiple bending operations as shown in Figure 1. The pipe bending process is complicated and forming conditions such as the control of load conditions are extremely important matter. For example, a lower axial load induced a reduction of thickness, while a lower pressure die load caused a wrinkle phenomenon at the bending corner as shown in Figure 2. To achieve a higher level of design, the practical use of CAE method is crucial. Therefore, we elaborated an analysis model for simple bending process.

In this paper, single bending process was examined. The analysis model consisted of one steel pipe deformable body and seven rigid bodies which were machine components. Abaqus/Explicit was used on this plastic deformation analysis which had complicated loading conditions as same as reality. In addition element technology was investigated. Several element types were applied on the same analysis condition, also machining conditions were studied.

All results were compared with the experimental data, and performance and accuracy were examined. Shell element showed good computation time but the reduction ratio of thickness was not provided, while the result of Incompatible solid element was as same as the experiment.

It is concluded that our proposal analysis methods can help us design more robust and provide the Machining condition determining method.

Figure 1. Exhaust manifolds.

Figure 2. 90 degree test.

2. Model Description

2.1 Parts and Assembly

Figure 3 and 4 showed an assemble FEA model which consisted of one deformable pipe and seven rigid parts. The model was analyzed by half model with symmetry boundary condition. The Deformable Pipe was simple circular shape and the thickness was 2[mm].

A real mandrel shape was shown in Figure 5. In this study, Mandrel model was made up of one long cylinder part and two bowls which were shown as red, green and white parts in Figure 3. These three parts were connected by the Join connecter condition between each reference points.

Contact pair was applied at each area. The coefficient of friction between Roll, Pressure, Wiper and outside pipe was 0.3, while the coefficient between Mandrel and inside Pipe was 0.1, because, lubricating oil was set in the pipe inside for smooth deformation.

First step, a rotation velocity condition was applied on the Bending Roll Die and the Clamping Die parts for bend forming. An axial certain load was applied on the left end of pipe to push the whole pipe toward X direction, while a historical pressure was put on the part of Pressure Die. These load conditions were argued below section 2.3 in detail. Also all degrees of freedom of the part of Wiper were fixed during the analysis. The X degree of freedom of the mandrel was fixed during first step in bending process.

Second step, after the bending process, the mandrel was withdrawn to minus X direction with a certain velocity.

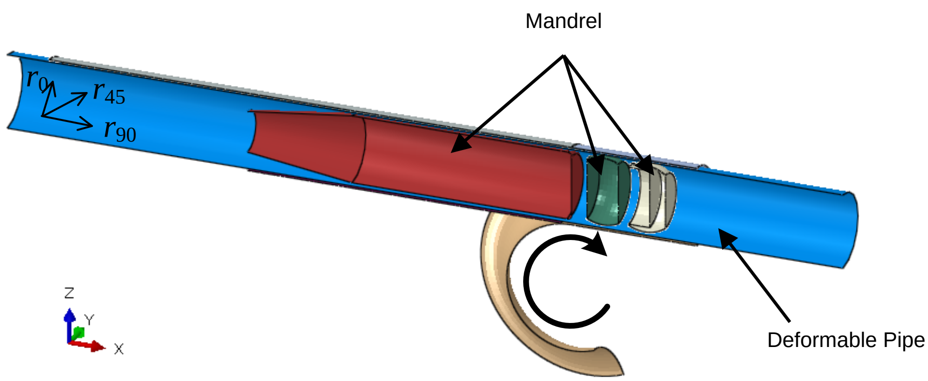


Figure 3. Assembly model.

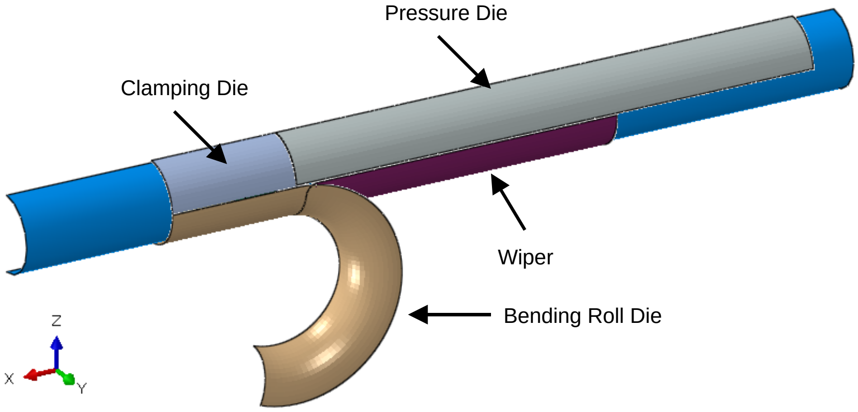


Figure 4. Assembly model.

Figure 5. Mandrel.

2.2 Material model

Material model of pipe was an elastic plastic behavior with anisotropic yield. The Young's modulus was 180[GPa], the Poisson's ratio was 0.3[-], and the density was 7.699E-6[Kg/mm^3].

Since the steel pipe was formed by rolling at first process, we should take into account anisotropic plastic flow. The r-value characterizes the anisotropic plastic behavior. For the planar anisotropic plastic, three parameters (r_0, r_{90} and r_{45}) which express the thickness reduction ratio of each direction are necessary. In our model, r_0 meant axial direction and r_{90} meant circumference, those were depicted in left side of Figure 3. These three parameters of r-value were experimented by Sango and the parameters of Potential option were calculated from Equation 1.

Figure 6 showed the stress strain curve, but these real values were confidential. Because of cyclic loading observed around bending area, nonlinear kinematic hardening was used. This hardening rule is well known method for sheet forming analysis.

$$R_{22} = \sqrt{\frac{r_{90}(r_0+1)}{r_0(r_{90}+1)}}, \quad R_{33} = \sqrt{\frac{r_{90}(r_0+1)}{(r_0+r_{90})}}, \quad R_{12} = \sqrt{\frac{3(r_0+1)r_{90}}{(2r_{45}+1)(r_0+r_{90})}} \tag{1}$$

$$R_{11} = R_{13} = R_{23} = 1.0$$

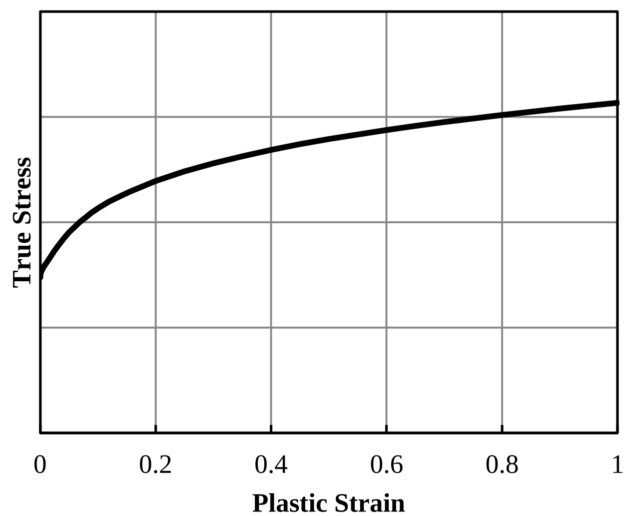


Figure 6. Stress-strain relation (Half cycle).

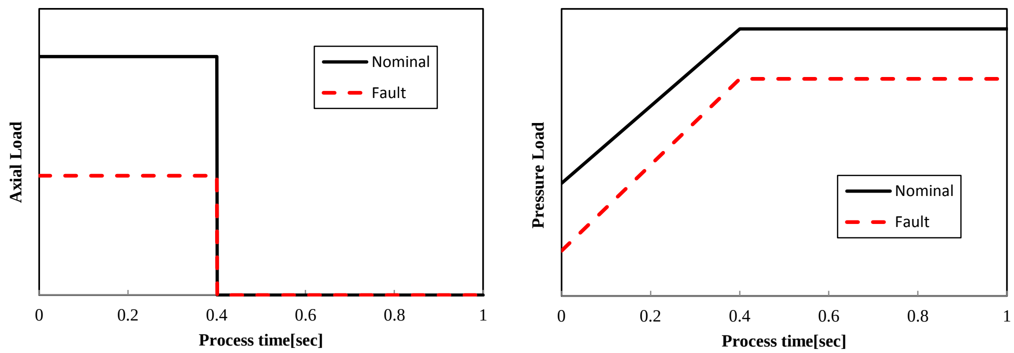
2.3 Load conditions

The load conditions are very important for bend forming. First condition, the Bending Roll Die and Clamping Die were applied an angle of 90 degrees rotation displacement in 0.4 seconds with certain velocity. Second condition, the axial load for Deformable pipe was applied as shown in Figure 7(a) toward global X direction. Third condition, the pressure die load for Pressure Die was applied as shown in Figure 7(b) toward global minus Z direction. The black solid line in Figure 7(a) and (b) showed typical nominal condition, the pipe was formed without any fault in those conditions. While, the breakage of thickness was observed in red dashed line in Figure 7(a), also, the wrinkle shown in Figure 2 appeared with the condition of red dashed line in Figure 7(b). To keep confidentiality, real values of vertical axis were inaccurate also.

Hence this study had three cases regarding load condition. The load cases were summarized in Table 1.

Table 1. Load case.

Load Case	Axial load	Pressure die load	Result of Experiment
a	Nominal	Nominal	Good shape
b	Lower	Nominal	Thickness reduction
c	Nominal	Lower	Wrinkle occurrence



(a) axial load of Deformable pipe **(b) Pressure die load**

Figure 7. Load controls.

2.4 Element types

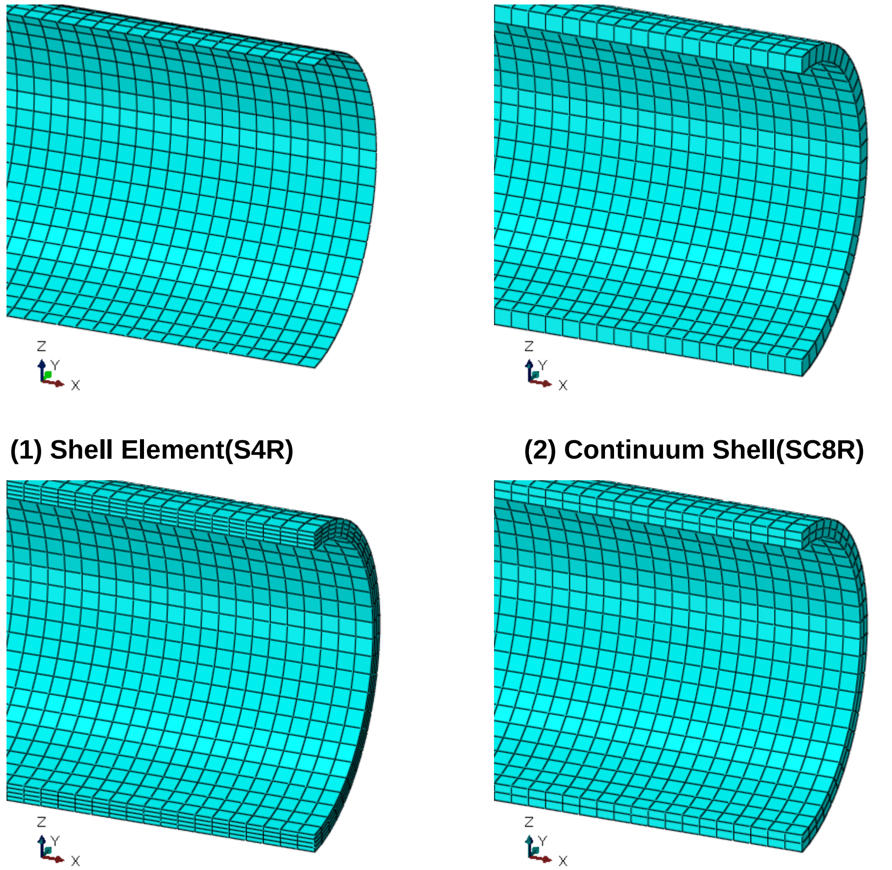
Needless to say, the character of bending behavior of element formulation is important in the process simulation, and our primary purpose was establishment of excellent estimation of thickness reduction and criterion of wrinkle occurrence. In this study, therefore, we examined various element types for the Deformable Pipe as shown in Figure 8. Also Table 2 summarized the element type, the number of layers of thickness, the number of nodes and elements.

As a consequence, we performed 12 cases of analyses of three load conditions and four element types.

Table 2. Element type.

Case	Element type	layers	Number of nodes	Number of elements	Note
1	S4R	5*	5829	5600	Shell element reduced integration
2	SC8R	1	11256	5400	Continuum shell element
3	C3D8R	4	28140	21600	Solid element reduced integration
4	C3D8I	2	16884	10800	Incompatible solid element

* The layers of shell element meant the number of integration points through the thickness.



(1) Shell Element(S4R) **(2) Continuum Shell(SC8R)**

(3) Reduced Integration Solid(C3D8R) **(4) Incompatible Solid(C3D8I)**

Figure 8. Element types and mesh controls of Pipe.

3. Results

3.1 Deformed shape

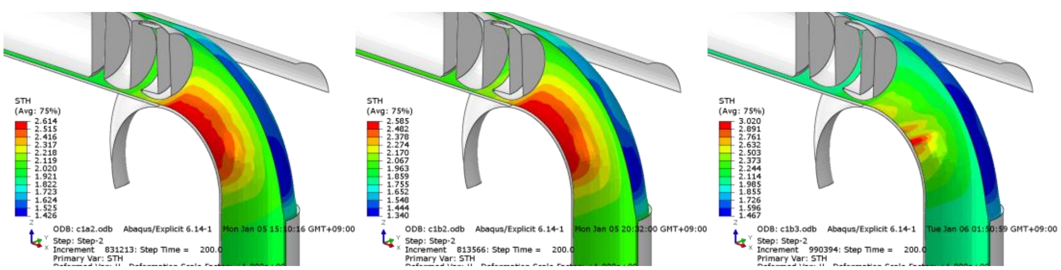
Figure 9 to 12 show the deformation shape and contour plot. Since element case 1 and 2 were structural shell element, the contour component was element thickness. While element case 3 and 4 were solid element, therefore, the contour plot showed the strain component of thickness direction.

Regarding to wrinkle occurrence which was appeared in load case (c), two types of S4R and C3D8R could not show the virtual wrinkle shape, while another types of SC8R and C3D8I revealed the wrinkle shape.

All element types could get the thickness reduction on the outer surface at bending area in the load case (a) and (b). Since the ratio of reduction is very important matter, Table 3 shows the minimum thickness at outer area on each analysis cases and experiment results on the last line. In case of Shell element, the ratio of S4R was as same as experiment in load case (a) within 0.14% relative ratio. But the ratio in load case (b) was 8.41%; it was not useful to estimate about the breakage. Also it could not reveal the wrinkle shape in load case (c). In the next case of Continuum shell element, SC8R showed larger thickness than experiment in both load cases (a) and (b), but it showed excellent wrinkle shape in load case (c). Third case of the reduced integration solid element, the relative ratio was 6.19% in load case (a) and 4.73% in load case (b). Apparently the element of C3D8R looked good result, however, it could not get the wrinkle shape in load case (c). The last case of the incompatible solid element C3D8I, the ratio in load case (a) was not good, but the ratio 1.46% in load case (b) was the best of all element types. Additionally it could show the good wrinkle shape in load case (c).

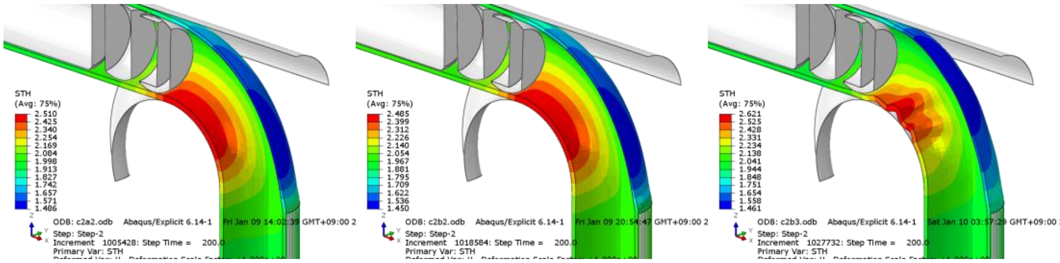
Table 3. Minimum thickness.

Element type	Load case	
	(a)	(b)
S4R	1.426(-0.14%)	1.340(+8.41%)
SC8R	1.486(+4.06%)	1.450(+17.31%)
C3D8R	1.340(-6.19%)	1.295(+4.73%)
C3D8I	1.327(-7.77%)	1.254(+1.46%)
Experiment	**1.428**	**1.236**



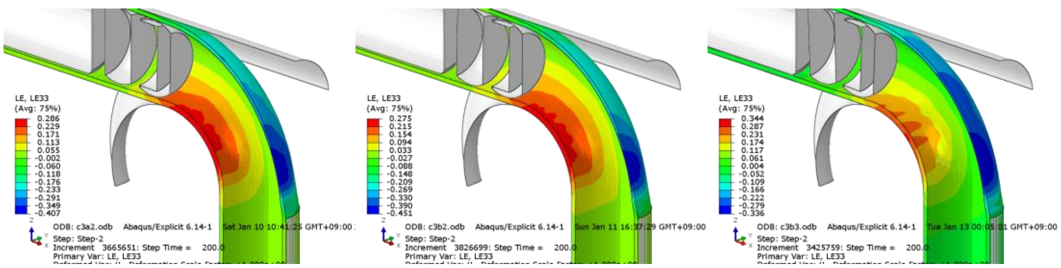
load case (a) **load case(b)** **load case(c)**

Figure 9. Thickness of Element type S4R.



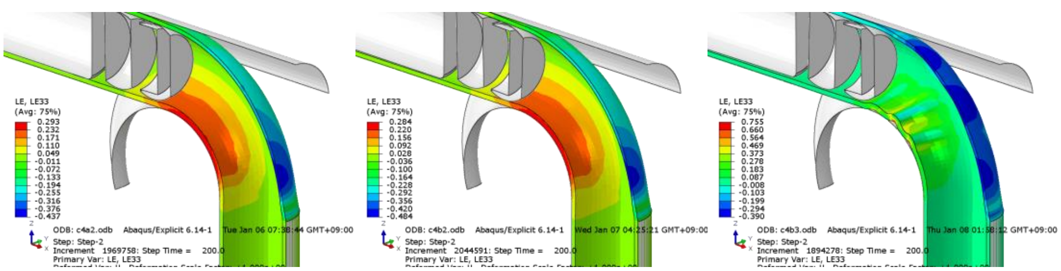
load case (a) load case(b) load case(c)

Figure 10. Thickness of Element type SC8R.



load case (a) load case(b) load case(c)

Figure 11. Strain of thickness of Element type C3D8R.



load case (a) load case(b) load case(c)

Figure 12. Strain of thickness of Element type C3D8I.

3.2 Analysis time

Table 4 shows the analysis time (wallclock time in the Abaqus log file) with parallel processing under Windows 7 64 bit operating system and a computer having two Intel Xeon E5-2620 (2.10GHz processor six cores). All jobs were analyzed under 12 cores parallel processing with Abaqus version 6.14-1.

Table 4. Analysis time.

Element type	Load case	Wall time [hh:mm]
S4R	a	3:53
	b	3:51
	c	4:15
SC8R	a	4:52
	b	5:00
	c	4:53
C3D8R	a	21:56
	b	22:48
	c	19:37
C3D8I	a	15:57
	b	16:34
	c	14:52

4. Discussion

In general, the shell element is traditional formulation for sheet metal forming, but this element could not show the large reduction of thickness in load case (b). The percentage relative to experiment was 8.41% which was insufficient to make a decision over breakage. Besides it could not reveal the wrinkle shape in load case (c). We assumed that these faults were caused from the difference of coefficient ratio of friction. Since pipe bending area had double side contact between the roll die and the mandrel, normal shell element could not handle shear deformation at cross section.

In the second approach, we tried using the continuum shell element which discretizes an entire three-dimensional body, unlike conventional shells which discretize a reference surface. This element is often used in case of double side contact. But this element could not treat the thickness reduction over 10%. We refer to one paragraph of Abaqus Analysis User's Guide 29.6.2 below and underline the important sentence.

"Although continuum shell elements discretize a three-dimensional body, care should be taken to verify whether the overall deformation sustained by these elements is consistent with their layer-wise plane stress assumption; that is, the response is bending dominated and no significant thickness change is observed (i.e., approximately less than 10% thickness change)."

As a result, the model of SC8R could not get enough the ratio of reduction in case (a) and (b), however, it showed superb wrinkle shape in case (c) for the sake of the advantage of double side contact.

In the third approach, the normal solid element with reduced integration was examined. In case of (b), the relative thickness was 4.73% which was not enough to decide the thickness failure in design process. The decision criteria is the most important in design process, therefore, this element was not adequate. Furthermore it could not show the wrinkle shape in case (c).

In the last approach, we have investigated the incompatible solid element which can handle the good bending behavior with few numbers of layers. Although the percentage of case (a) was 7.77%, the most important reduction ratio of case (b) was in 1.46% relative, this precision was the best of all cases. Moreover the wrinkle shape was arisen from the double side contact with different friction ratios on both sides of the pipe. But the large computing cost was shortcoming even in using only two layers as shown in Table 4.

In conclusion, we studied various element types in pipe bending simulation. To acquire the thickness reduction and the occurrence of wrinkle, we found that the incompatible solid element is the best way for the pipe bending formulation. However this element cost large amount of computing resources. Therefore Sango usually analyze with the shell element in normal design, however, they use the incompatible solid element in case of detail and precise design.

5. References

1. Abaqus Analysis User’s Guide, Version 6.14, Dassault Systèmes Simulia Corp., Providence, RI.