Mechanical Behavior of a Pipe-in-Pipe in a Span

Theofilos Giagmouris, Xinhai Qi, Andy Tang, Jianxia Zhong

Genesis, North America

Abstract: In a free span, the inner pipe of a pipe-in-pipe system (PiP) bends substantially due to internal pressure and temperature and possibly contacts the outer pipe. The study is motivated by the need to understand how the interaction between the two pipes affects the PiP behavior at a free span during thermal expansion and how a PiP can be modeled adequately. The pipes were simulated with thick wall pipe, elbow, and shell elements. The interaction between the inner and outer pipes was simulated with multi-point constraints (MPC), tube-to-tube contact elements (ITT), and contact surfaces. Finite element (FE) model accuracy and cost effectiveness were compared in a case study.

Keywords: Pipe-in-pipe, Free span, Pipeline, Bending

1. Introduction

A PiP is shown in Figure 1. In a free span, Figure 2, the pipes bend and the gap between them allow for relative displacement due to different effective tension and weight of the pipes. A shell FE model of a PiP at a span was developed to investigate the behavior during operating and cool down conditions. In addition, pipe FE models were developed and evaluated against the shell FE model. It is shown that while the shell FE model simulates the motion of the inner pipe relative to the outer pipe and the contact between them, it requires a lot of computational effort. Contrary, a pipe FE model provides global deformation and pipe stresses close to those of the shell FE model and is cost effective. Table 1 and Table 2 summarize the PiP properties, span under investigation, and loading conditions for each pipe.

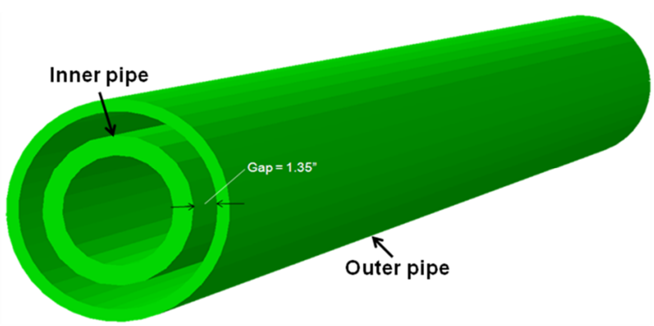

Figure 1. Pipe-in-pipe system.

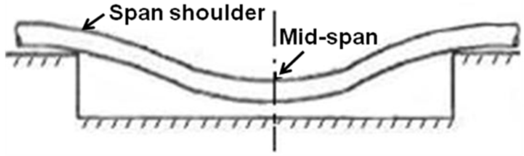

Figure 2. Pipe at a symmetric span.

Table 1. Geometric and material properties of the PiP.

Pipe	Outer Diameter D (in)	Thickness t (in)	D/t	Steel Grade	Elastic Modulus (ksi)	Poisson's Ratio	Thermal Expansion Coefficient (1/°F)	Symmetric Span Length (ft)
Outer	12.750	0.709	18.0	API 5L X65	29700	0.3	$6.4 \cdot 10^{-6}$	120
Inner	8.625	1.142	7.6					

2. Shell element model

A shell FE model was developed in Abaqus/Standard, to simulate a PiP at a symmetric span (120 ft length) allowing relative motion and contact between the pipes. The model is symmetric about an axis that passes through the mid-span (left-hand-side edge of the pipes in Figure 3) and about a vertical plane that passes through the axis of the PiP reducing the domain to one-fourth by applying symmetric conditions. "Hard" surface-to-surface contact was adopted, with the inner surface of the outer pipe acting as the "master" surface and the outer surface of the inner pipe as the "slave" surface assuming a friction factor μ=0.3. Both pipes were discretized with 4-node shell elements with reduced integration and hourglass control (S4R). 10 elements were used around the 180° circumferential sector for the inner pipe and 20 for the outer pipe, 120 elements were used along the length.

Table 2. Loading conditions for each pipe.

Pipe	Loading Condition: As-laid / Cool down	Loading Condition: Operation
Outer	2.1 ksi external hydrostatic pressure	
	40° F steel temperature	
	40.9 lbf/ft weight (adjusted for the correct submerged weight of the PiP)	
Inner	0 ksi internal pressure	9.3 ksi internal pressure
	40° F steel temperature	235° F steel temperature
	94.2 lbf/ft weight (no contents)	108.0 lbf/ft weight (with contents)

The inner pipe is concentric initially with the outer pipe. Both pipes at the span shoulder (right-hand-side edge of the pipes in Figure 3) are constrained by not allowing vertical movement and are free to rotate around the lateral axis.

Figure 3 shows a PiP at a span during the as-laid condition. The PiP was loaded by the weight of the pipes and external hydrostatic pressure at the outer pipe. Bending occurs at both pipes due to their weight. The gap between the inner and outer pipes results in relative displacement. During operation, the internal pressure, the steel temperature, and the weight of the contents were applied at the inner pipe. As shown in Figure 4, contact occurs at the lower half of the pipes during operation. The same stress contour limits were used for the as-laid and operating conditions. After cool down both pipes are at ambient temperature and the inner pipe is empty.

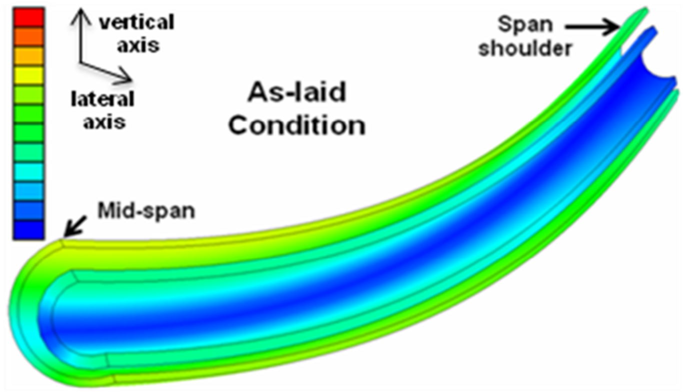

Figure 3. von Mises stress contours of a PiP in as-laid condition.

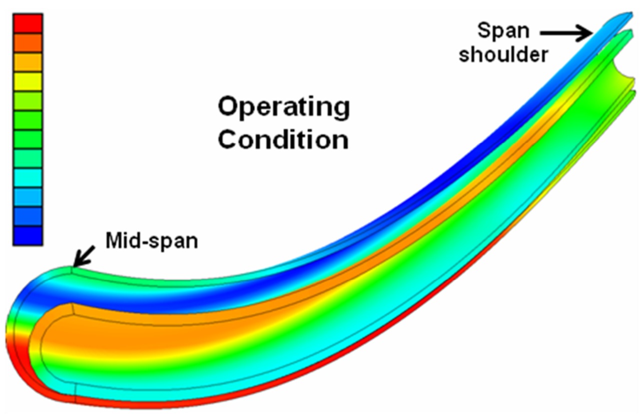

Figure 4. von Mises stress contours of a PiP in operating condition.

To model the global behavior of a PiP at a span location under thermal expansion and contraction, an FE model was developed which accounts for a larger PiP section that extends “far away” from the span as shown in Figure 5. The model includes a 2000 ft flat seabed which allows modeling the interaction between the outer pipe and the seabed and a 60 ft curved seabed section simulating a symmetric span (120 ft length). The seabed was modeled as an analytical rigid surface.

The interaction between the outer pipe and the seabed was modeled as “soft” contact, with the seabed acting as the “master” surface and the outer surface of the outer pipe as the “slave” surface. The model assumed an embedment 50% of PiP outer diameter under the PiP submerged weight and an axial soil friction factor μ=0.46. Symmetry condition was applied at the mid-span. “Far away” from the span (left-hand-side edge in Figure 5) the two pipes were tied, by making the end displacements equal, simulating a bulkhead.

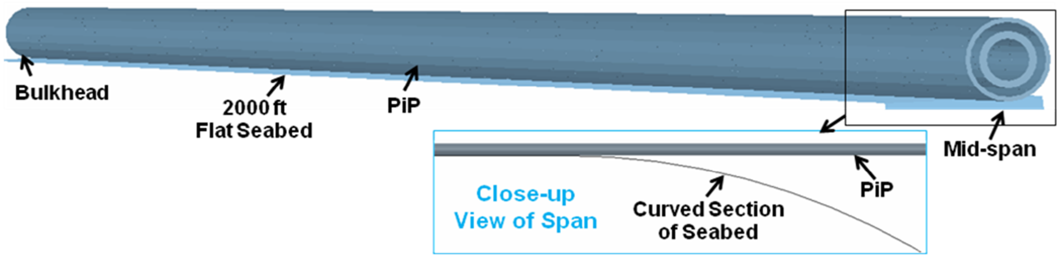

Figure 5. PiP model including a flat seabed with a curved end section to simulate a span.

3. Pipe element models

The global behavior of the PiP at the span was also simulated by three different pipe FE models. This allowed for a comparison of the maximum pipe stress, downward displacement at mid-span and computational time between the pipe FE models as well as the shell FE model. For all models, the same geometry, boundary conditions, loading, and material properties were applied. The four models (the three pipe FE models and the shell FE model), along with the type and number of elements used, are shown in Table 3 and listed as follows:

1. Pipe elements assuming no radial relative displacement between the pipes (MPC)
2. Pipe elements (for minor or non bending sections) and elbow elements (for significant bending sections) assuming no radial relative displacement between the pipes
3. Pipe elements using tube-to-tube contact elements to simulate contact with friction between the pipes (ITT)
4. Shell elements assuming contact with friction between the pipes, the model is also symmetric about the central axis of the PiP

Table 3. FE models and discretization of the geometry.

FE Model	Element Type	Total No. Elements	No. Elements "away from span" each pipe	Element Length "away from span"	No. Elements "span area" each pipe	Element Length "span area"
1. MPC	PIPE31H	2780	910	2 ft	480	0.5 ft
2. Elbow	PIPE31H ELBOW31	2720 60	910 -	2 ft -	450 30	0.5 ft 0.5 ft
3. ITT	PIPE31H ITT31	2780 2780	910 910	2 ft 2 ft	480 480	0.5 ft 0.5 ft
4. Shell	S4R	25320	3640 (inner) 7280 (outer)	5 ft, 10 at 180° 5 ft, 20 at 180°	4800 (inner) 9600 (outer)	0.5 ft, 10 at 180° 0.5 ft, 20 at 180°

Note: "span area" is considered as 240 ft from mid-span and "away from span" is the rest 1820 ft of the PiP section

3.1 Pipe Element Model with MPC

The PiP was modeled using 2-node linear hybrid pipe elements (PIPE31H) which account for axial stretch, bending and hoop stress due to internal/external pressure at the material constitutive calculations. This assumption leads to a geometrically simple model. A constraint (slider MPC) keeps the inner pipe on the axis of the outer pipe but allows it to move along the axis. By using this constraint, the two pipes always remain concentric under thermal expansion and contraction.

The thin-walled formulation, where the hoop stress is assumed constant through the cross section and the radial stress is neglected, is valid for $D/t > 20$. Since both pipes have $D/t < 20$, the thick-walled formulation is used for the pipe elements. At the thick-walled formulation the hoop and radial stress vary through the thickness and are calculated based on Lame's equations.

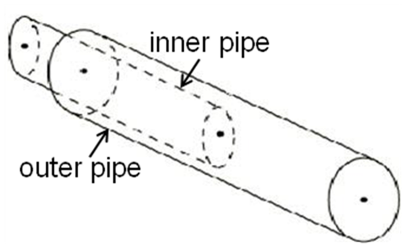

Figure 6. Schematic of a PiP where the pipes remain concentric during deformation.

3.2 Pipe and Elbow Element Model with MPC

For cases involving pipes exhibiting large amounts of bending, ovalization can occur and result in weak behavior. Ovalization may be an issue for a PiP at the mid-span. The assumption of pipe elements is that the cross section cannot deform in its own plane. Elbow elements can model the deformation through the pipe wall and provide accurate modeling when ovalization is substantial.

Elbow elements (ELBOW31) were used at the mid-span area, and pipe elements (PIPE31H) were used for the rest PiP section. A constraint (slider MPC) was also used in the pipe and elbow element model to keep the two pipes concentric during deformation.

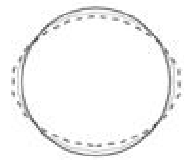

Figure 7. Initial (solid line) and deformed (dashed line) cross section of an elbow element.

3.3 Pipe and Tube-to-Tube Contact Element Model

Tube-to-tube contact elements can model the interaction between two pipes where one lies inside the other. For the PiP, a set of tube-to-tube contact elements (ITT31) was used with pipe elements (PIPE31H) to include interaction between the pipes. “Hard” contact was adopted with a friction factor μ=0.3 (same interaction properties were used as in the shell FE model).

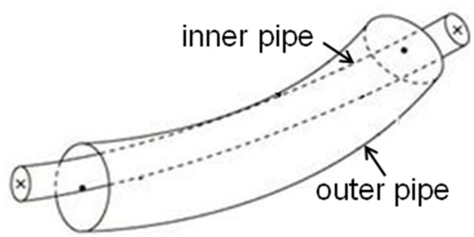

Figure 8. Schematic of an inner pipe contacting the outer pipe using pipe and tube-to-tube contact elements.

4. Comparison of FE Models

The deformed configuration of the PiP at the span during operating and cool down conditions is illustrated in Figure 9 and Figure 10 respectively. As expected, the PiP embeds significantly at the span shoulder during operation. A close-up view of the PiP deformed configuration during operation is shown in Figure 11. An embedment of 50% of the PiP outer diameter at the area “far away” from the span and large embedment at the span shoulder were observed for all models.

The deformation of the PiP at the mid-span for all models during operation and cool down is illustrated in Figure 12 and Figure 13 respectively. During operating and cool down conditions the global response for all models was similar as shown in Figure 9 and Figure 10 respectively. However, in the pipe element model (MPC) the inner pipe remains concentric with the outer pipe while, in the ITT and shell element models, the inner pipe contacts the outer pipe at the lower half. The model with the pipe and elbow elements is not illustrated as it has the same deformation with the pipe element model (MPC).

Figure 9. PiP deformed configuration at span during operating condition.

Figure 10. PiP deformed configuration at span during cool down condition.

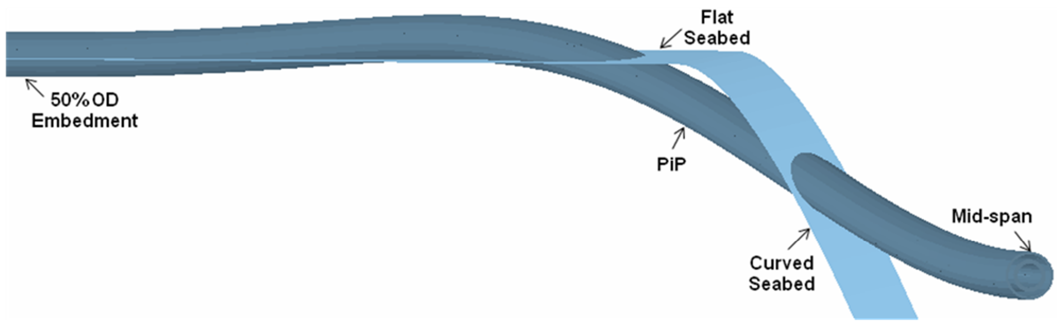

Figure 11. Close-up view of the PiP deformed configuration at span during operation.

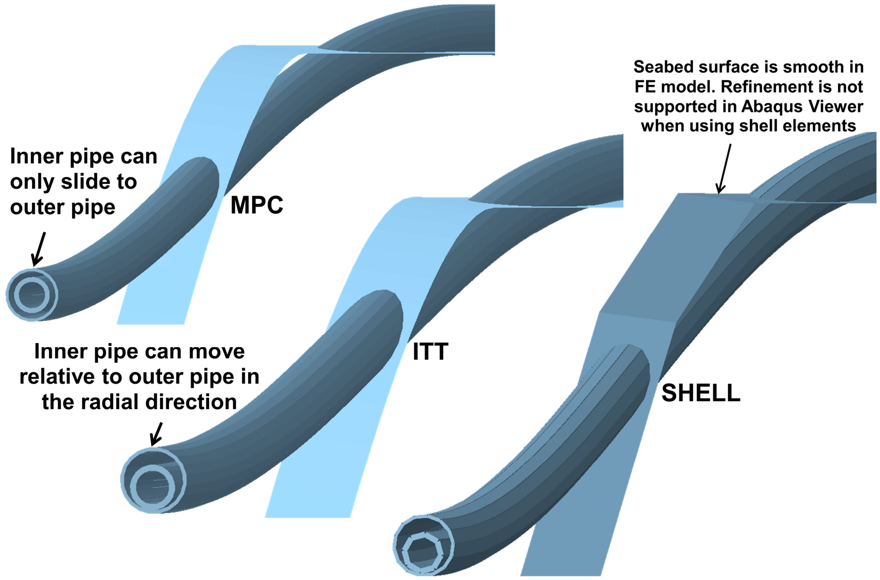

Figure 12. PiP deformation at mid-span during operating condition for MPC, ITT and shell FE models.

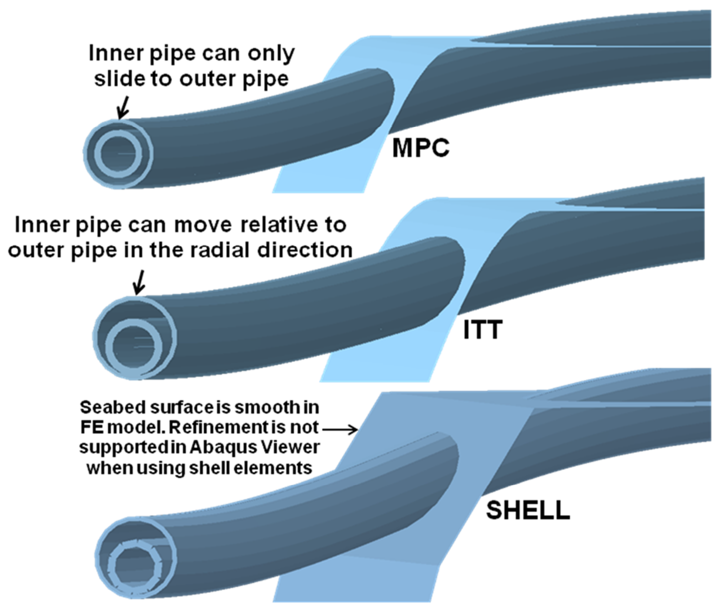

Figure 13. PiP deformation at mid-span during cool down condition for MPC, ITT and shell FE models.

Depending on the approach adopted every model predicts different downward displacement at mid-span ($\delta_{mid\text{-}span}$) and maximum pipe (von Mises) stress at the pipe mid-wall (σ_{vM}) as shown in Table 4. The predicted displacement at mid-span is similar for all models for operating and cool down conditions since the overall deformation of the PiP was similar. For all models, the maximum pipe stress occurs at the outer pipe mid-span area. Pipe ovalization is not significant and contact between the pipes (simulated by ITT and shell FE models) does not predict significant stress increase in the contact area.

The model with the pipe and elbow elements predicts higher pipe stress compared to the MPC model by considering the effect of pipe ovalization in the free span. The shell FE model accounts for pipe ovalization and contact including friction between the two pipes. For the span under investigation, the maximum pipe stress from the MPC model is fairly close to that of the shell FE model. Note that the MPC model neither contemplates the pipe nor the friction between two pipes. The ITT model which simulates the interaction between the two pipes predicts a maximum pipe stress similar to that of the shell FE model.

The longitudinal stress range ($\Delta\sigma_L$) at the pipe mid-wall between operating and cool down conditions for each model is also shown in Table 4. The stress range values are similar for all models. Fatigue calculations based on Paris law suggest that the number of cycles to failure at constant stress range ($\Delta\sigma$) is proportional to $\alpha/(\Delta\sigma)^m$ where α, m are the constant and gradient of an S-N curve, respectively. The difference in the number of cycles to failure relative to the ones predicted based on the shell FE model longitudinal stress range and for an S-N curve with m=3 is shown in the last column of Table 4.

Table 4. Downward displacement at mid-span and maximum pipe stress during operation and cool down.

	Loading Condition				**Fatigue Damage Calculations**	
	Operation		**Cool down**			
FE Model	**$\delta_{mid\text{-}span}$ *(in)***	**Max σ_{vM} *(ksi)***	**$\delta_{mid\text{-}span}$ *(in)***	**Max σ_{vM} *(ksi)***	**$\Delta\sigma_L$ *(ksi)***	**Difference on predicted number of cycles to failure (%)**
1. MPC	66.5	59.2	26.4	20.4	44.7	- 8
2. Elbow	65.5	62.5	26.7	21.7	46.1	- 1
3. ITT	62.4	60.2	26.7	19.7	47.2	+ 0
4. Shell	66.7	59.8	27.2	21.0	46.9	–

Table 5 lists the computational time and the ease with which the model provided a solution. Values relative to the MPC model are also provided. Models able to simulate contact between the two pipes (ITT and shell FE models) require substantial computational time and can have serious convergence issues which can often make them undesirable for PiP modeling.

Table 5. Computational time and convergence.

FE Model	Absolute CPU Time	Relative CPU Time	Convergence
1. MPC	392 s	1	Easy
2. Elbow	468 s	1.2	Easy
3. ITT	4197 s	10.7	Difficult – penetration and contact force issues
4. Shell	63899 s	163	Very Difficult – penetration and contact force issues

5. Conclusions

The following preliminary conclusions can be drawn from the Abaqus simulations:

- Elbow elements should be used when ovalization is significant (severe bending location). If a PiP does not experience substantial bending, elbow elements may not be required.
- Shell elements model the motion of the inner relative to the outer pipe and account for contact between them. However, interaction between the two pipes generally does not lead to excessive pipe stress increase in the contact area. In addition, shell elements are computational expensive.
- Similar to shell elements, tube-to-tube contact elements used with pipe elements allow for interaction between the two pipes but also require high computational effort.
- Pipe elements with a constraint to keep the two pipes concentric during deformation lead to a simple model. In cases where bending and friction between the two pipes is not significant, this pipe element model is reasonably accurate and relatively cost effective for modeling a PiP at the conceptual design.
- The conclusions listed above would also be applied for PiP sections with bending due to other causes than spanning, such as for lateral buckling of a PiP laid on the seabed and upheaval buckling of a buried PiP.

6. Acknowledgments

Appreciation is extended to OTC. Parts of this paper are copyright 2013, Offshore Technology Conference. Reproduced with permission of OTC, further reproduction prohibited without permission.

7. References

1. Abaqus Users Manual, Version 6.12-1, Dassault Systémes Simulia Corp., Providence, RI.

2. Giagmouris, T., Qi, X., Tang, A., Zhong, J., “Response and Simulation of the Pipe-in-Pipe in Free Spans,” Offshore Technology Conference, OTC-24073-MS, 2013.