# Finite Element Analysis of the Local Strength of the Insertion Sleeve Structure for the Lifting Frame of Long Span Jacket

Jingyi Cao(✉), Xin Bian(✉), and Huaizhou Huang

CNOOC Offshore Oil Engineering Co., Ltd., Tianjin 300451, China
`cjy19851115@126.com`, `13072961083@163.com`

**Abstract.** The lifting of the long span "8 + 4" leg split jacket requires the use of a specific lifting frame, where the local structure of the lifting frame insertion sleeve is the key part of the lifting, and it is extremely necessary to calculate and analyze it. This study focuses on a large span "8 + 4" leg jacket and its lifting frame structure in China. A lifting model of the "8 + 4" leg jacket was established using SACS software and its overall calculation and analysis were conducted. However, due to the limited computing power of SACS software for local structures, this study extracts relevant boundary conditions of the target local structure of the model based on the overall calculation and analysis of SACS, and combine ABAQUS to model, calculate, and analyze the local structure of the insertion sleeve, in order to provide reference for the engineering site.

**Keywords:** Lifting of large-span jacket · Lifting frame · Insert point sleeve structure · ABAQUS · Local structure

## 1 Introduction

The offshore installation methods for jackets are mainly divided into lifting and launching installation methods and sliding launching installation methods. In ultra shallow water areas, the lifting and launching installation method is generally used. The development of the lifting and launching installation design plan needs to be based on the weight and size of the jacket, the total height of the lifting rigging, and the lifting weight and lifting capacity range of the main operating vessel [1].

A domestic "8 + 4" leg jacket will be lifted and installed in a sea area with a water depth of 10.5 m, with a distance of 68 m between the centers of the "8" and "4" leg jackets. For the lifting and installation of this ultra shallow water long span "8 + 4" leg jacket, a specific lifting frame needs to be used [2], and the plug-in sleeve structure is one of the most dangerous areas of the lifting frame. The main reason for this is that during the lifting operation, as an important load-bearing structure connecting the jacket and the lifting frame, this area is affected by structural mutations and stress concentration, resulting in high local stress, There is a risk of strength failure.

The design calculation of the large span "8 + 4" leg jacket mainly uses SACS software as the analysis and verification software. However, the software's calculation

B. Sun et al. (Eds.): DWOG-Hyd 2023, LNCE 472, pp. 182–189, 2024.
https://doi.org/10.1007/978-981-97-1309-7_17

and analysis capabilities for local structures such as lifting ears and lifting frame spigot sleeves are not very comprehensive. Therefore, based on the overall calculation results of SACS for the “8 + 4” leg jacket and its lifting frame, a finite element model for the local structure of the lifting frame spigot sleeve was established using the finite element method and calculated and analyzed in this study.

## 2 Calculation of Jacket and Lifting Frame Based on SACS

### 2.1 Introduction to the “8 + 4” Leg Jacket and Lifting Frame Structure

The structure of a domestic “8 + 4” leg jacket and its lifting frame is shown in Fig. 1. The lifting points are set inside the 2 and 3 axis pile legs, and the lifting height requirement is about 62.5 m. The elevation of the working point is 7.5 m, and the dimensions of the 4-leg and 8-leg jacket working points are both 14 m × 20 m, lifting frame working point size is 40 m × 20 m, with a water depth of 10.3 m, and the elevations of each horizontal layer are EL (+) 5.5 m, EL. (−) 2.5 m, and EL. (−) 9.8 m.

### 2.2 SACS Model of “8 + 4” Leg Jacket and Lifting Frame Structure

This article uses the marine engineering professional software SACS for modeling and calculation, and the final SACS model established based on geometric dimensions is shown in Fig. 2.

In the SACS model, the hook is completely fixed and its degree of freedom is set to 111111. Since the lifting rope only bears axial loads, the degree of freedom is set to 011111. The lifting calculation adopts a 10% amplification factor for the dead weight of the jacket and its lifting frame structure by modifying the density of the structural group, resulting in a final dead weight of 2430 tons.

The overall calculation mainly considers two working conditions, namely, the components and joints directly embedded in the lifting lug should be calculated according to the minimum load factor of 2.0, while the minimum load factor of all other components and joints should be calculated according to 1.35. The minimum load factor of 2.0 is the maximum local stress in the plug-in sleeve structure, and the plug-in sleeve structure under this working condition is selected as the local analysis object.

### 2.3 Boundary Extraction of Models in SACS

Due to the complex structure of the lifting frame insertion sleeve, and the incomplete calculation and analysis of this local structure by SACS, a simplified simulation was conducted on the insertion sleeve part of the lifting frame during the overall modeling stage to provide corresponding boundary conditions for subsequent finite element calculations. Among them, the XY direction release was performed on the joint part of the insertion point and sleeve, and the simplified model of the insertion point sleeve structure in SACS is shown in Fig. 3a.

This article uses the large-scale finite element calculation software ABAQUS to calculate and analyze the lifting frame spigot sleeve structure. Due to the fact that SACS

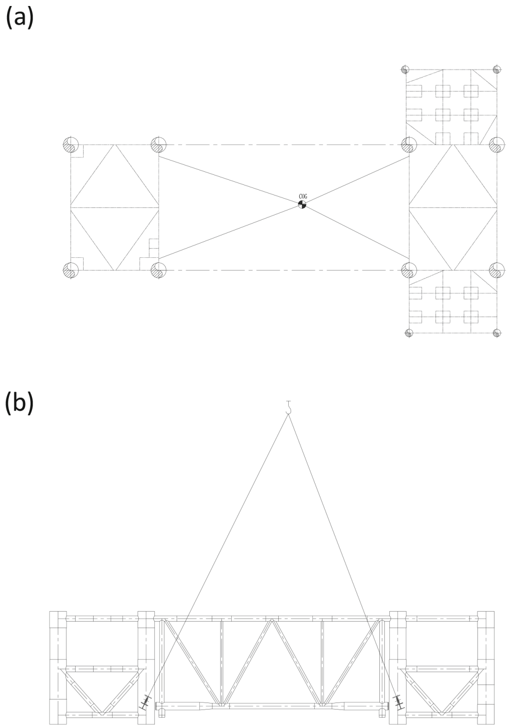

**Fig. 1.** The “8 + 4” leg jacket and lifting frame structure: (a) Top view; (b) Front view

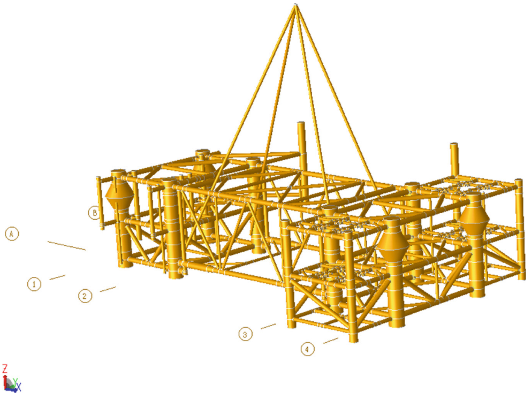

**Fig. 2.** SACS model of the “8 + 4” leg jacket and its lifting frame structure

software uses a local coordinate system to represent the direction and magnitude of forces, while ABAQUS software generally uses a global coordinate system, it is necessary to extract node forces from the SACS local coordinate system and apply them to

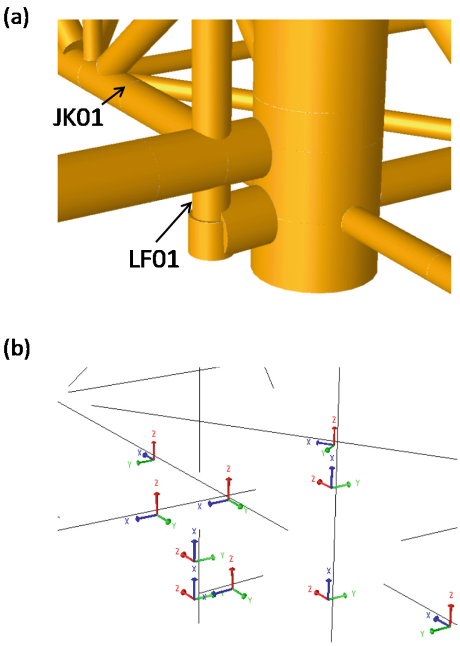

**Fig. 3.** (a) SACS Simplified Model of Lifting Frame Insert Sleeve Structure; (b) Local Coordinates of SACS Simplified Model with Pointed Sleeve Structure

the finite element model. The simplified SACS model with a spigot sleeve structure has local coordinates as shown in Fig. 3b [3].

**Table 1.** Forces and Bending Moments at the Cutoff Point

| Forces and bending moments | Joint | |
|---|---|---|
| | LF01 | JK01 |
| Fx/KN | 0 | 3617.64 |
| Fy/KN | −6400.76 | 31.34 |
| Fz/KN | −21.67 | −13.72 |
| Mx/(KN·m) | 0 | 73.75 |
| My/(KN·m) | 7.94 | −14.32 |
| Mz/(KN·m) | 1935.26 | −18.86 |

Considering that in the whole jacket and its lifting frame model, the stress at the junction of the spigot sleeve mainly comes from the extrusion of the side near the jacket spigot against the inner wall of the sleeve, and the stress at the junction of the slant support connected to the sleeve and the column connected to the main leg is also large, in order to facilitate the application of loads and boundary conditions in the ABAQUS local finite element model, The force of the frame member at the cut-off point LF01 directly connected to the spigot is extracted as the extrusion force of the side of the spigot against the inner wall of the sleeve (as shown in Fig. 3a), and the force at the member cut-off point JK01 between the jacket A and B axes is extracted as the applied load at the junction of the slant support connected to the sleeve and the column connected to the main leg. The forces and bending moments extracted from LF01 and JK01 points are shown in Table 1.

## 3 Calculation of Insert Sleeve Structure Based on ABAQUS

### 3.1 Finite Element Model

The ABAQUS model of the spigot sleeve structure is shown in Fig. 4a, which mainly includes several parts such as the truncated jacket pile leg and partial structure, the lifting frame spigot, sleeve, ring plate, and horizontal tie bar through the bottom, all of which use a 4-node shell element S4R. After modeling all the above components, select the TIE constraint in ABAQUS to assemble all the components into a complete model. Considering the interaction between the insertion point and the sleeve during the operation process, in order to simulate this situation more realistically, the contact position is set as node surface contact, normal hard contact, and tangential friction coefficient of 0.3, as shown in Fig. 4b [4].

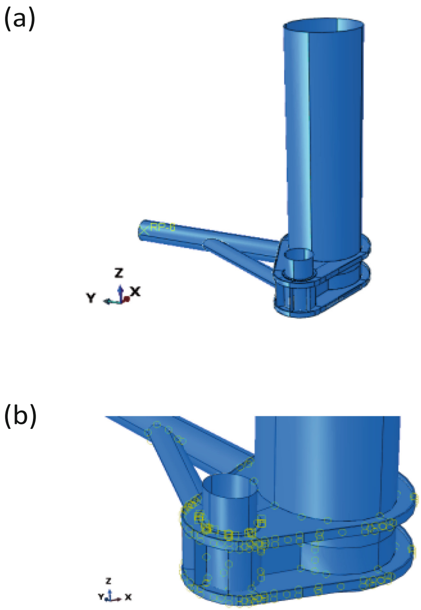

**Fig. 4.** (a) Finite element model of lifting frame spigot sleeve structure; (b) Interaction

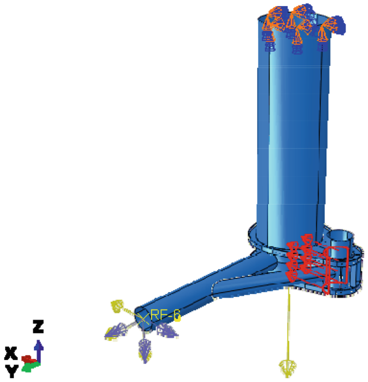

**Fig. 5.** Load and Boundary Conditions of Insert Sleeve Structure

Due to the fact that the forces of the bars extracted in the SACS model are all extracted at the cut-off point of the bar where they are located, which corresponds to the structure in ABAQUS software but is a circular space structure in the form of shell elements. Therefore, in order to accurately apply the forces of the cut-off point in SACS to the circular space structure in ABAQUS, it is necessary to set an RP point and set a Coupling constraint with the cross-section of the circular space structure, as shown in Fig. 4a.

### 3.2 Load and Boundary Conditions

As previously described, the force on the local structure of the spigot sleeve mainly comes from the self weight of the structure, the squeezing force on the inner wall of the sleeve caused by the side of the lifting frame spigot, and the internal force of the members between the A and B axes of the jacket. The self weight of the local structure is achieved by setting the steel density and applying global gravity acceleration, and fully constraining the top of the pile leg, as shown in Fig. 5.

Due to the fact that the insertion point cannot fully contact the inner surface of the sleeve during the operation, in order to simulate the stress situation at the junction of the insertion point and the sleeve more realistically, the extrusion force on the inner wall of the sleeve from the insertion point side of the lifting frame is applied in the form of surface load to the inner wall of the sleeve. The value is Fy at node LF01 divided by 1/2 of the surface area of the inner wall of the sleeve, which is 2.1 Mpa.

Apply the force and bending moment at the cut-off point JK01 between the A-axis and B-axis of the jacket extracted by SACS at reference point RP-6. As the SACS software uses a local coordinate system to represent the direction and magnitude of the force, ABAQUS software generally uses a global coordinate system. The final force and bending moment applied at RP-6 in ABAQUS are shown in Table 2.

### 3.3 Stress Results and Analysis

As shown in Fig. 6, the Mises stress distribution cloud map of the spigot sleeve structure during lifting operations. Under the action of the structural self weight, the side of the

**Table 2.** Forces and bending moments applied in ABAQUS

| Forces and bending moments | Reference point |
|---|---|
| | RP-6 |
| Fx/N | 31340 |
| Fy/N | 3617640 |
| Fz/N | −13720 |
| Mx/(N·m) | −14320 |
| My/(N·m) | 73750 |
| Mz/(N·m) | −18860 |

lifting frame tip will form a squeezing force on the inner wall of the sleeve, which is transmitted to the pile legs through the ring plate. The maximum stress in Fig. 6 occurs at the connection between the lower ring plate and the pile leg, with a maximum value of 164 MPa. From the calculation results, it can be seen that the Mises stress in various parts of the local structure of the plug-in sleeve meets the strength requirements of the material, indicating that the strength of the plug-in sleeve can meet the requirements of the design conditions.

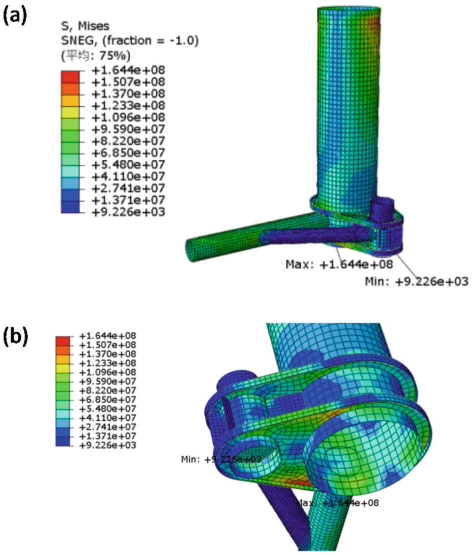

**Fig. 6.** Mises stress distribution diagram of inserted sleeve structure

## 4 Conclusion

This article uses SACS software to establish an "8 + 4" leg jacket lifting model and conducts overall calculation and analysis. Based on this, combined with ABAQUS, the local structure of the spigot sleeve is modeled, calculated, and analyzed, and the following conclusions are drawn:

(1) In the SACS modeling stage, in order to provide corresponding loads and boundary conditions for the subsequent finite element calculation of the local structure of the lifting frame insertion sleeve, it is necessary to simplify the simulation of the insertion sleeve part of the lifting frame, and release the insertion and sleeve joint part in the X and Y direction.
(2) Due to the inconsistency between SACS and ABAQUS calculation software in representing the magnitude and direction of rod forces, it is necessary to accurately extract node forces in the SACS local coordinate system, and correctly apply forces and boundary conditions in the ABAQUS local model.
(3) In order to simulate the stress situation between the insertion point and the sleeve more realistically, the contact analysis method was used to simulate the contact state between the insertion point and the inner side of the sleeve.
(4) Considering that during the operation process, the insertion point cannot fully contact the inner side of the sleeve, a pressure of 1/2 area on the inner side of the sleeve is selected.
(5) After calculation, the Mises stress of each part of the local structure of the plug-in sleeve meets the strength requirements of the material.

## References

1. Jiang, M.: Offshore Engineering Structures: Jacket Platforms (2009)
2. Xiao, H., Bo, Z.: Installation plan analysis for ultra-shallow split jacket in JZ 9-3 oilfield. Ocean Eng. Equip. Technol. **3**(03), 164–167 (2016)
3. Chen, B.M., Yu, C.J., Wang, Z.C., et al.: Finite element analysis of local strength of jacket skirt sleeve. Ship Ocean Eng. **47**(04), 148–152+156 (2018)
4. Shi, D.D.: Engineering Application Example Set of ABAQUS Software (2015)