点击复制

大型海洋平台基础结构有限元分析(PDF)

Title: Finite element analyses of large-size offshore platform foundation

作者: 徐秀丽[1]; 张元壮[1]; 刘伟庆[1]; 潘晓峰[1]; 王仁贵[2]

1. 南京工业大学 土木工程学院, 江苏 南京 210009;

2. 交通部 公路规划设计院, 北京 100010

Author(s):   XU Xiu-li[1]; ZHANG Yuan-zhuang[1]; LIU Wei-qing[1]; PAN Xiao-feng[1]; WANG Ren-gui[2]

1. College of Civil Engineering, Nanjing University of Technology, Nanjing 210009, China;

2. Highway Planning and Design Institute, Ministry of Communications of China, Beijing 100010, China

摘要: 某大型海洋平台基础采用平台板+桩基础组合形式,承台面为椭圆形,台面尺寸为147.6m×99.0 m,平台板为混凝土板和钢梁组成的叠合梁板结构。该工程地处世界三大强潮海湾之一,所受荷载与当地地质条件均较复杂。介绍了应用有限元对其进行结构分析的实施方法及过程,采用弹性抗力法考虑了桩-土共同作用,重点探讨了钢梁加混凝土厚板组成的叠合梁的有限元模拟方法。通过对比分析,提出了计算结果精度高、可操作性强的叠合梁有限元模拟方案。有限元分析结果表明,该平台钢管混凝土桩与钢梁均满足材料强度要求,但在施工阶段,桩内存在拉力。

Abstract: A large-size offshore platform with assembled platform slab-pile foundation has an elliptic shape platform of 147.6 m×99.0 m dmiension.The platform slab is a superposed beam-slab structure constructed by steel beam and concrete slab.The gulf where the platform locates is one of the three biggest tide gulfs in the world,and the geological condition there is complex.The structure bears complex load.In this paper,a process of static ana lysis for this structure was introduced by finite element method.The pile-soil interaction problem was solved by elastic-resistance method,and the simulation of composite steel beam-concrete slab was mainly discussed.The analysis method proposed in this paper is

high-precise and high-operable.The finite element analysis result shows that the material strength demand of CFST (concrete filled steel tubes)-piles and steel-beams can be satisfied,but there is pull in the piles on the stocks.

参考文献/REFERENCES

-

更新日期/Last Update: 1900-01-01