

## 《单词百科:NSY是什么意思?NSY怎么发音?NSY的解释和用法》

英语单词NSY是什么意思?NSY怎么读?NSY怎么发音?简答网为您整理了NSY的解释、用法、例句、词组等相关学习资料。下面跟小编一起来看看吧!



### NSY的意思

1、abbr.New Scotland Yard 新苏格兰场;neoxanthin synthase 新甲藻黄素合酶;normal speaking younger children 年幼儿童正常说话;nylon textured yarn 尼龙变形纱线;

### NSY的双语例句

1、To thermal field , ANSYS can highly approach nonlinear transient heat conduction problem.

ANSYS可以高度逼近非线性瞬态热分析问题,并且有处理相变的能力.

2、The fatigue critical site aside turbine disk pin-hole was located based on the FEA of ANSYS.

通过ANSYS有限元分析得到涡轮盘危险点应力应变分布,确定疲劳危险部位.

3、 In comparison with the results of ANSYS, FEA software , the results of the programs are satisfying.

并将分析结果与大型有限元分析软件ANSYS分析所得结果作了对照,得到了较好的一致性.

4、 The results of ANSYS numerical simulation validate that the cable support technique can control entry stability.

ANSYS数值 模拟结果证明,该锚索补强支护方案对巷道围岩控制是有效的.

5、 The stress , displacement and modality of the hyperboloidal shell is analysed, with the help of ANSYS 9 ? 0.

以配煤仓落料斗为对象,在ANSYS9 . 平台上对双曲 料仓斗壳进行了静载计算和模态分析.

6、 The modal analysis of a machine tool was done by means of software ANSYS.

采用了ANSYS软件对机床模态特性进行了有限元分析.

7、 AIP ( Air Independent Propulsion System ) system provides effective solutions for power system of conventional submarine.

不依赖空气的推进系统,即AIP ( AirIndependentPropulsionSystem ) 系统的出现为常规潜艇的动力系统提供了有效的解决方案.

8、 This thesis analyzes the carrying force mechanism of SRC column by ANSYS software.

本文用ANSYS软件分析了型钢混凝土柱的受力机理.

9、 The burst of soft rock is studied by program ANSYS.

ANSYS程序分析了权台煤矿软岩冲击矿压.

10、 Second , carries on the wind load static computation using ANSYS to the silo model.

利用ANSYS对筒仓模型进行风载静力计算.

11、 Engineering Applied Tutorial of ANSYS ( Mechanical ) [ M ]. Beijing: China

Railway Publishing House, 2003.

ANSYS工程应用教程 ( 机械篇 ) [ M ]. 北京: 中国铁道出版社, 2003.

12、 With ANSYS, This paper applies the trussed beam as supporting beam of trays in big-diameter-distillation tower.

本文应用ANSYS分析计算, 成功地在精馏塔内采用了桁架梁结构作为大直径精馏塔塔盘支撑.

13、 The rock mass stress and tectonic stress zones are analyzed with ANSYS Program.

文中用ANSYS软件 结构静力分析方法,对岩体应力作了分析研究,划分了构造应力区.

14、 Based on on the base of ANSYS, the author designs calculation programs, upbuilds three-dimensional DP elastic-plastic apply model to carry on nonlinear simulation analysis.

在此基础上, 基于ANSYS平台, 编制了计算程序, 建立三维DP弹塑性有限元计算模型, 进行非线性仿真分析.

15、 Base on the basic theory of finite-element analysis, hypostatic modeling for four-side simple supporting plank was upbuilt with ANSYS software. Three-dimensions hypostatic model and plane model were present by simulating the strain distribution of four-side simple supporting plank with damage using finite-element analysis software.

介绍了有限元分析的理论基础, 用ANSYS软件对四边简支板试件进行了实体建模, 比较了三维实体模型和平面模型特点, 并指出了采用平面模型进行建模的合理性;

16、 Modeling Studying of TI Worm in the Environment of ANSYS

基于ANSYS环境下TI蜗杆的造型研究

17、 Finite element software ANSYS was employed to investigate the mechanical response of asphalt pavement structure under the axisymmetrical dynamic loading, and different loading period, loading amplitude and loading form were discussed in detail.

运用有限元程序ANSYS, 采用轴对称模型, 求出了沥青路面在动荷载作用下的力学响应, 并针对不同的荷载作用周期、荷载幅值以及不同的荷载作用形式进行了详细的分析与讨论。

18、 To compute axile rigidity by using ANSYS, can get various deformation for multi-section exactly and simultaneously.

利用有限元分析软件ANSYS计算轴的刚度，可以准确地同时得出多个截面的多种变形。

以上是简答网为您整理的NSY怎么读的相关信息，希望对大家有一定的帮助。查看更多关于NSY的用法、NSY的释义、NSY的相关详情请点击：<https://dict.jiandongshicai.cn/NSY>

