[Local Stress Prediction] User Manual

Table of Contents

---

1. Introduction

利用有限元方法计算压力容器喷嘴连接处局部应力的过程十分复杂。为了减轻分析的难度，开发了基于 ABAQUS 和 ML 的预测软件。该软件能够快速预测压力容器喷嘴连接处的局部应力，精度水平超过 0.999，均方误差仅为 1.639。这种方法与传统的有限元分析设计方法形成对比，因为它绕过了许多复杂的分析步骤。并且我们还提供了软件开发的方法和需要的工具，这将帮助用户开发特定工况和材料参数下的预测软件。它为快速评估和优化压力容器设计提供了一个可靠、方便的平台，同时也为不同工况和材料下的应力预测提供了参考。

2. Product Overview

The Local Stress Prediction package comprises an ABAQUS script and a corresponding plugin (Generate Dataset) for generating datasets, Python code for machine learning and model generation, and a comprehensive stress prediction software (Stress Prediction-ML). This means that users can refer to these publicly available files to create stress prediction software tailored to specific working conditions and materials.

One ABAQUS script and one plugin are publicly available and serve the purpose of generating datasets. This script or plugin can create nozzle models in batches based on user-provided parameters and subsequently calculating the stress within them. All the calculated data is saved in a .txt file, which can be transformed into datasets in various formats as per requirements. Modifying the script or plugin allows users to obtain datasets for different materials and working conditions, although it is recommended to limit modifications to material parameters and avoid altering other parameters.An example of the dataset is also provided here, which has 2860 samples (date-1.scv).

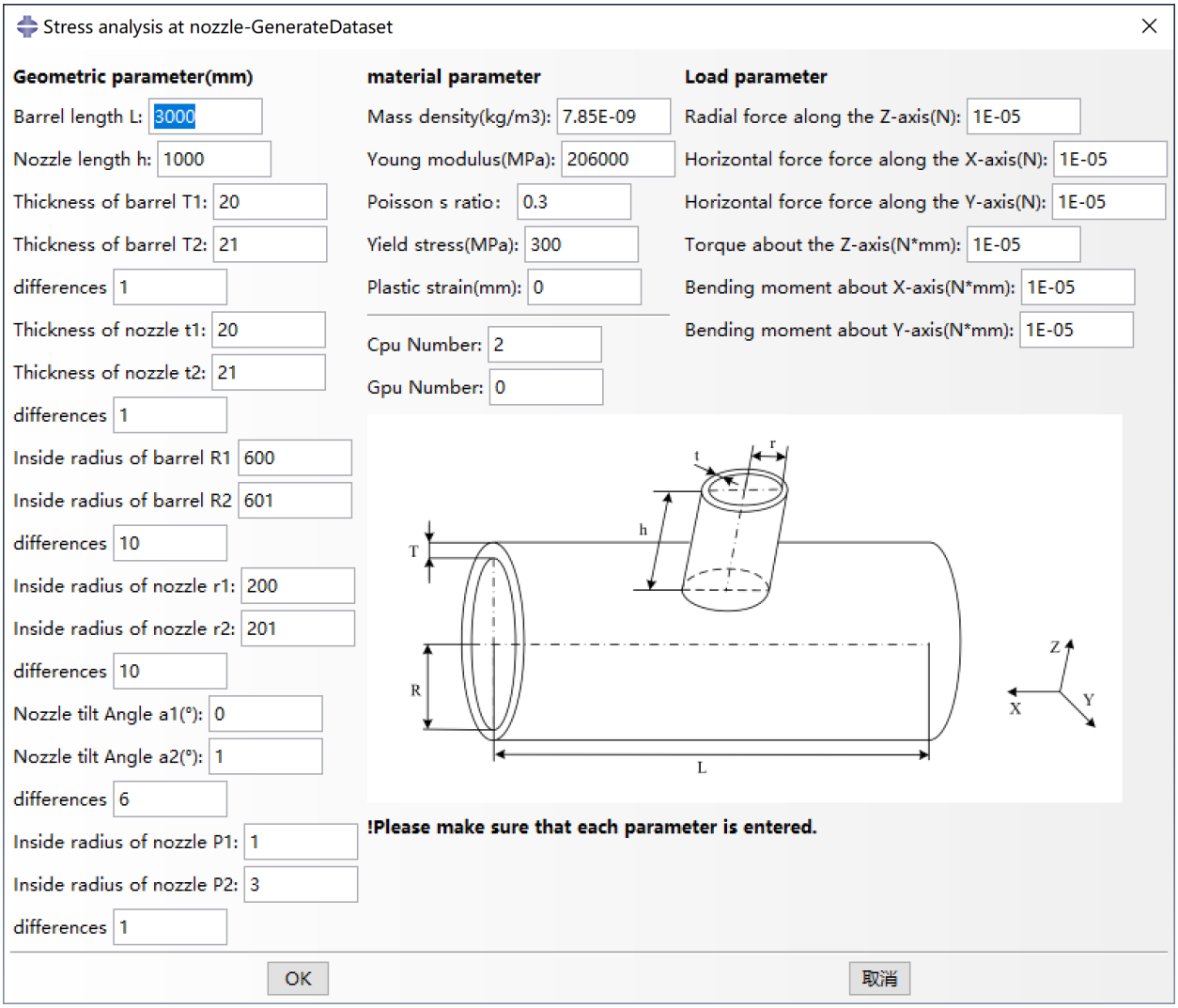
After generating the dataset, machine learning can be performed using Python code to create personalized predictive models. This enables the generation of stress prediction models for any working conditions and materials. These models are designed to carry out predictive tasks, and

users have the option to develop graphical user interfaces (GUIs) to provide fully-fledged software that can be used by non-technical individuals.
The complete stress prediction software is an illustrative example that can be directly utilized. It employs Q345 as the material and assumes normal temperature as the working condition. On top of this, it uses parameters such as pressure and geometry as features for machine learning to derive the software's predictions.

3. Installation/Setup Instructions

GenerateDataset

1. Copy the "GenerateDataset" folder to 'C:\Users\Username\abaqus_plugins'.
2. Open ABAQUS, and you will find "GenerateDataset" under the "Plug-ins" menu.
3. Open "GenerateDataset" and enter the parameters. Click "OK" to complete the batch analysis.
4. In the folder "C:\temp\Local_pipe_analysis", you will find the "MaxMises.txt" file.
5. Copy or convert the data into the desired format for your dataset.



Stress_Prediction-ML

1. Press Win+R to open the Run dialog box, then type "cmd" and press Enter. This will open the Command Prompt window.
2. In the Command Prompt window, enter the following commands (install one after another once the previous one is installed): pip install pyqt5 pip install pandas pip install scikit-learn pip install matplotlib pip install seaborn
3. Double-click on "Stress_Prediction-ML.exe" to run the program. Enter the corresponding geometric parameters and click OK to obtain the predicted values.
4. You can copy and save the results for each set of input parameters for further analysis.

Stress Prediction-ML

T: 12

t: 12

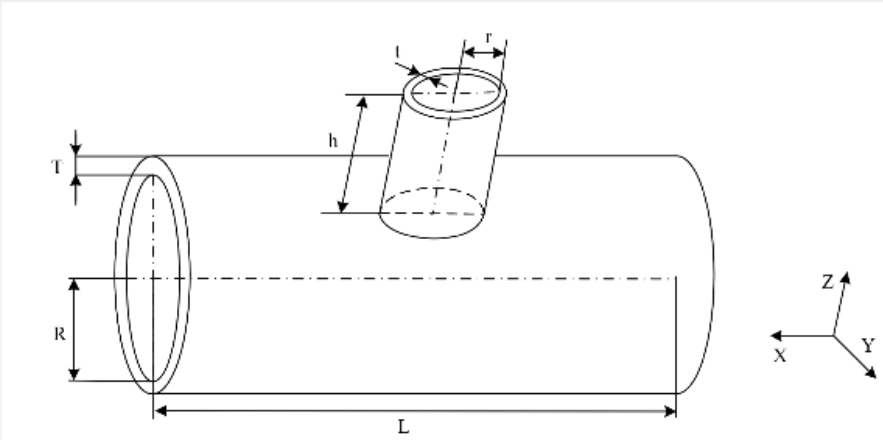
R: 300

r: 120

a: 10

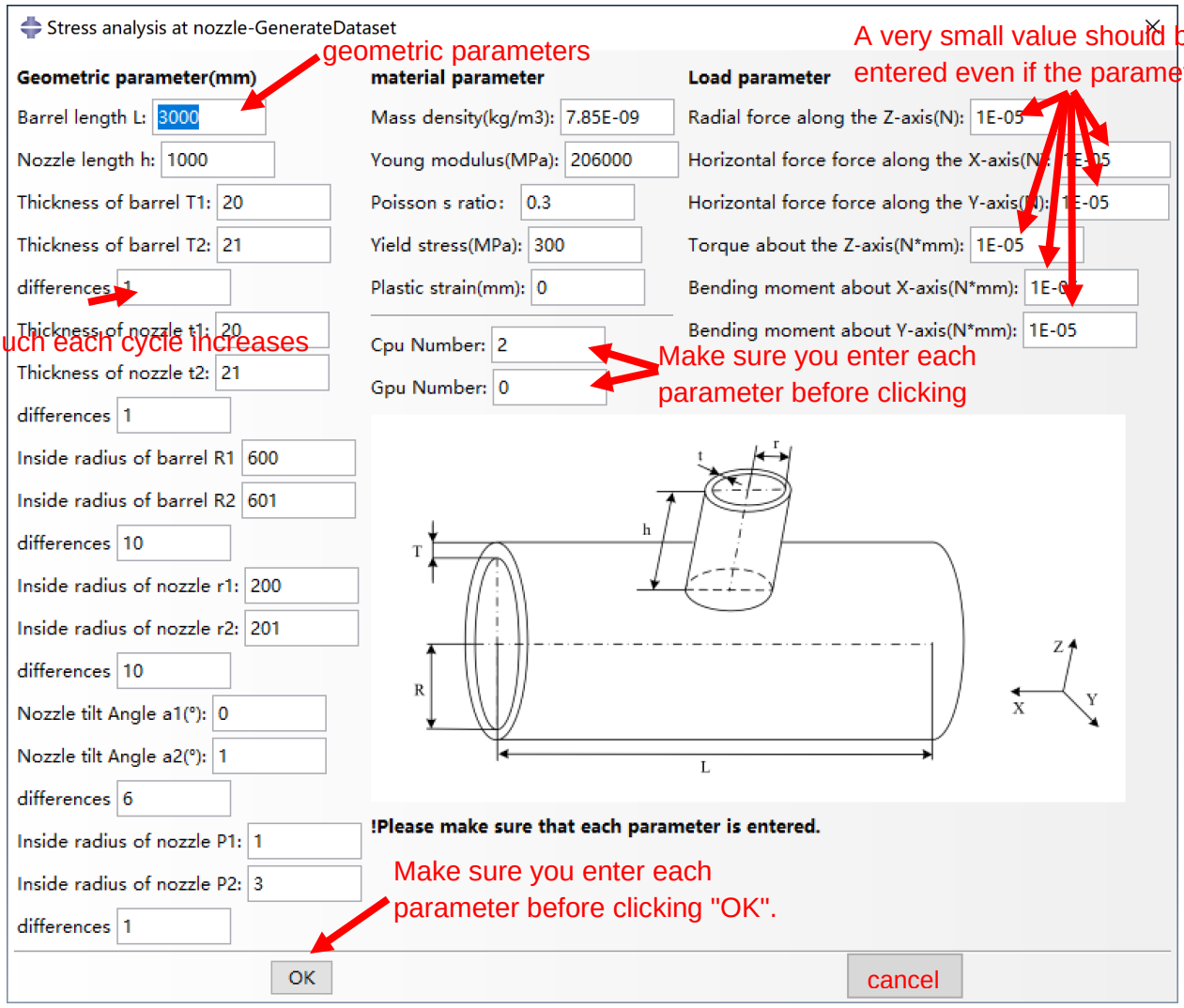
P: 0. 5

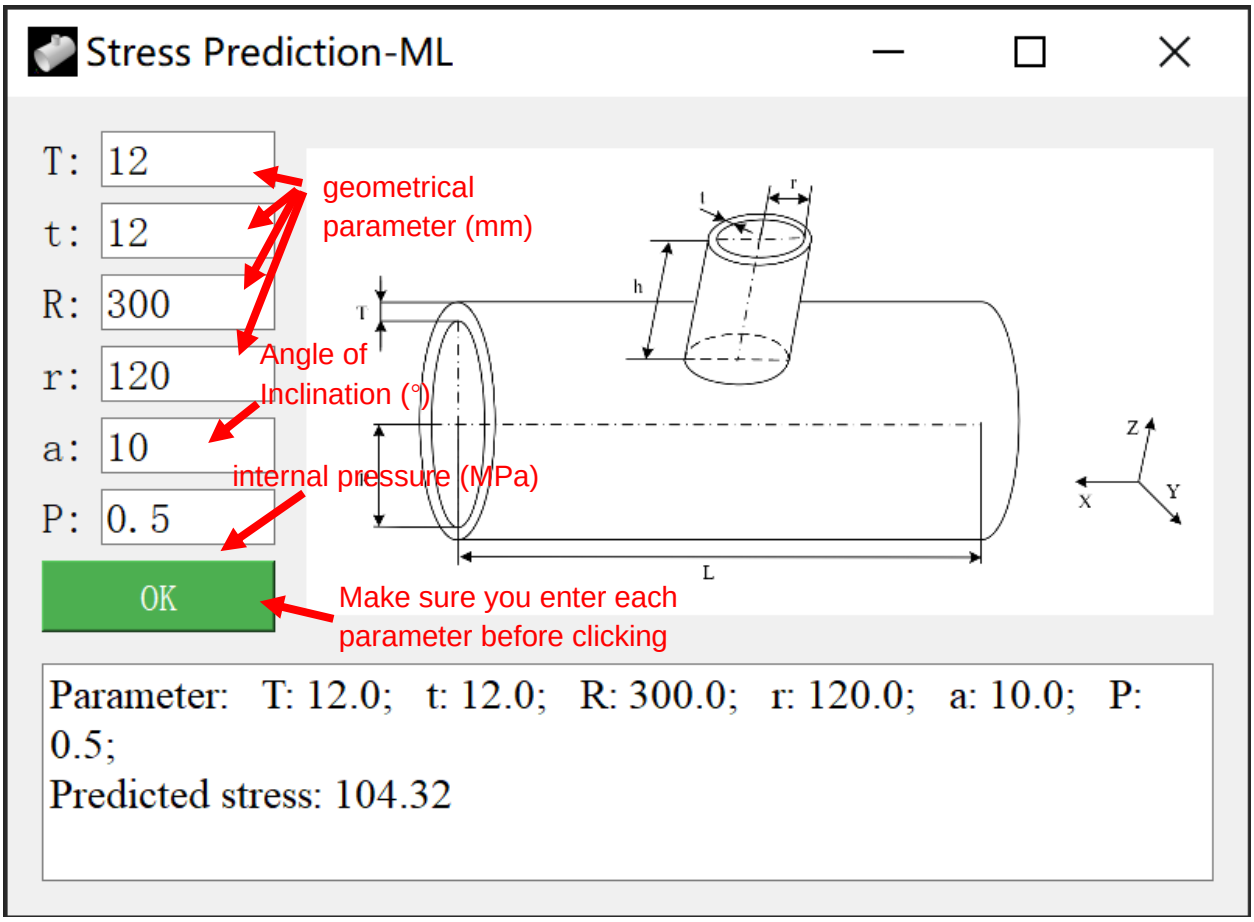
OK



Parameter: T: 12.0; t: 12.0; R: 300.0; r: 120.0; a: 10.0; P: 0.5;
Predicted stress: 104.32

4. Operating Instructions





5. Troubleshooting

如有问题请留言，或邮件:Fanhangchao@163.com

6. Disclosures

详情参考 Licence.txt。