

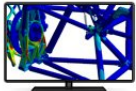
1 Introduction

This document and example file are only used for academic purpose and released in MIT License. We demonstrate how to use batch mode with Python script to change the ANSYS Workbench's parameters (or variables), execute the simulation and export results to text file.

The basic concept of the integration for your program with other softwares should be similar.

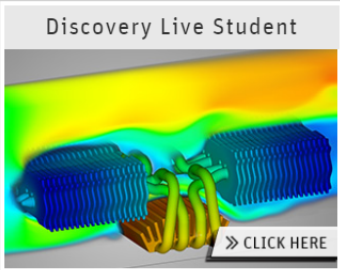
2 Install ANSYS Student

We are **not** encourage students to use commercial softwares without license. The ANSYS Student Software is provided a twelve-month renewable license and limited with 32k nodes/elements in Structure Analysis, 512k nodes/cells in Fluid Analysis. You can download it from the website : <https://www.ansys.com/academic/free-student-products>




ANSYS Free Student Software Downloads

ANSYS provides free student software products perfect for work done outside the classroom, such as homework, capstone projects, student competitions and more. Our renewable products can be downloaded by students across the globe. ANSYS Student products can be installed on any supported MS Windows 64-bit machine. Learn more about Discovery Live Student, ANSYS AIM Student and ANSYS Student so you can download them today.



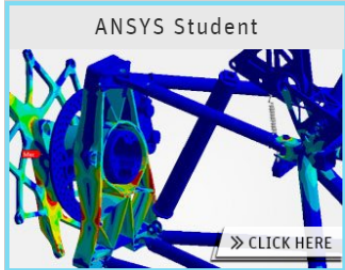
Discovery Live Student

» CLICK HERE



ANSYS AIM Student

» CLICK HERE




ANSYS Student

» CLICK HERE

▼ Download ANSYS Student 19.2

ANSYS Student is our ANSYS Workbench-based bundle of ANSYS Mechanical, ANSYS CFD, ANSYS Autodyn, ANSYS SpaceClaim and ANSYS DesignXplorer. ANS students globally. It is a great choice if your professor is already using it for your course or if you are already familiar with the ANSYS Workbench platform.

[DOWNLOAD ANSYS STUDENT 19.2](#)



2

Figure 1: ANSYS Free Student Software Downloads

3 Example

3.1 Download

Download the example file from NTU CEIBA or GitHub : <https://github.com/solab-ntu/batch-ansys-workbench> .The file **example.zip** should contain at least 4 items which include following files/folder:

- model_v192_student.wbpj
- model_v192_student_files/
- batch_run_ansys.py
- batch_cmd.bat

The project file (model_v192_student.wbpj) should only be opened in ANSYS Student v19.2 or newer version.

3.2 Problem Description

In this model, we construct a cantilever beam which has three sections as shown in Figure 2. Each section has a circle sketch and uses the diameter size as the design variables.

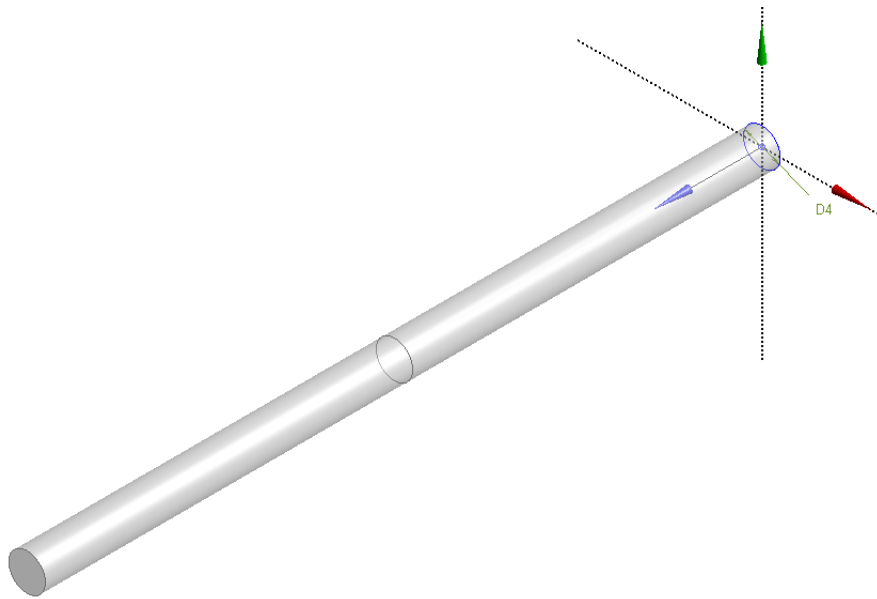


Figure 2: Geometry

In structural simulation, we fixed one end of the beam and apply a 100 [N] force (+x) on the other end. The stress and deformation distribution are shown in Figure 3. Then we can retrieve the response values to Parameter Set Table, as shown in Figure 4.

Optimization in Engineering

Appendix - Run ANSYS Workbench in batch mode

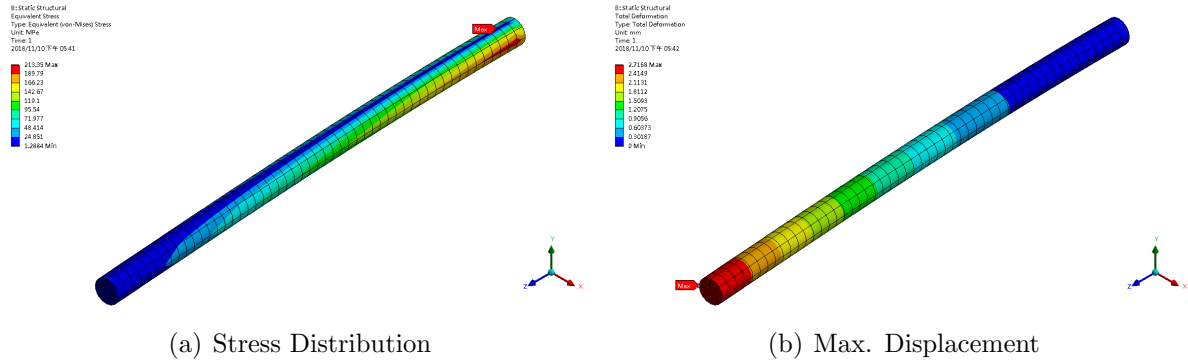


Figure 3: Analysis Result

Outline of All Parameters				
	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Geometry (A1)			
4	P1	XYPlane.D4	10	mm
5	P2	Plane4.D1	10	mm
6	P3	Plane5.D1	10	mm
*	New input parameter	New name	New expression	
8	Output Parameters			
9	Static Structural (B1)			
10	P4	Solid Volume	15708	mm ³
11	P5	Equivalent Stress Maximum	213.35	MPa
12	P6	Total Deformation Maximum	2.7168	mm
*	New output parameter		New expression	
14	Charts			

Figure 4: Parameters

3.3 Batch Mode

- batch_run_ansys.py

Before running the batch mode. You need to check the project file's location in the script. This file path might be different in your computer.

```
1 # encoding: utf-8
2 # -- open wbpj
3 Open(FilePath="E:/temp/model_v192_student.wbpj")
```

Then you can decide which parameter should be changed and modify the value by the "Expression" argument.

```
5 # -- setup parameter value
6 p1 = Parameters.GetParameter(Name="P1")
7 p2 = Parameters.GetParameter(Name="P2")
8 p3 = Parameters.GetParameter(Name="P3")
9
10 p1.SetQuantityUnit("mm")
```

```
11 p2.SetQuantityUnit("mm")
12 p3.SetQuantityUnit("mm")
13
14 dp0 = Parameters.GetDesignPoint(Name="0")
15 dp0.SetParameterExpression(Parameter=p1, Expression="10")
16 dp0.SetParameterExpression(Parameter=p2, Expression="10")
17 dp0.SetParameterExpression(Parameter=p3, Expression="10")
```

After updating the project. We can use Python file I/O function to write out the parameter's information to a plain text file.

```
22 # — write out result
23 fileIO = open("E:/temp/output.txt","w")
24
25 for parameter in Parameters.GetAllParameters():
26     value = parameter.Value.ToString()
27     fileIO.write(parameter.Name + ", " + value + "\n")
28     fileIO.flush()
29
30 fileIO.close()
```

- batch.cmd.bat

```
1 "D:\Program Files\ANSYS Inc\ANSYS Student\v192\Framework\bin\Win64\
  RunWB2.exe" -B -R "batch_run_ansys.py"
```

The first part of this .bat file is the path of ANSYS WB program, and the argument means running the program with batch_run_ansys.py in batch mode. These two files (batch_run_ansys.py and batch.cmd.bat) should be placed in the same folder.

After double clicking batch.cmd.bat. It will appear two MS-DOS windows as shown in Figure 5, and create output.txt when it finished. The output.txt will contain the parameter's information of the project file.

For the detailed operation procedure, please check the video :
<https://www.youtube.com/watch?v=jGWYmR0uqtU>

Optimization in Engineering

Appendix - Run ANSYS Workbench in batch mode

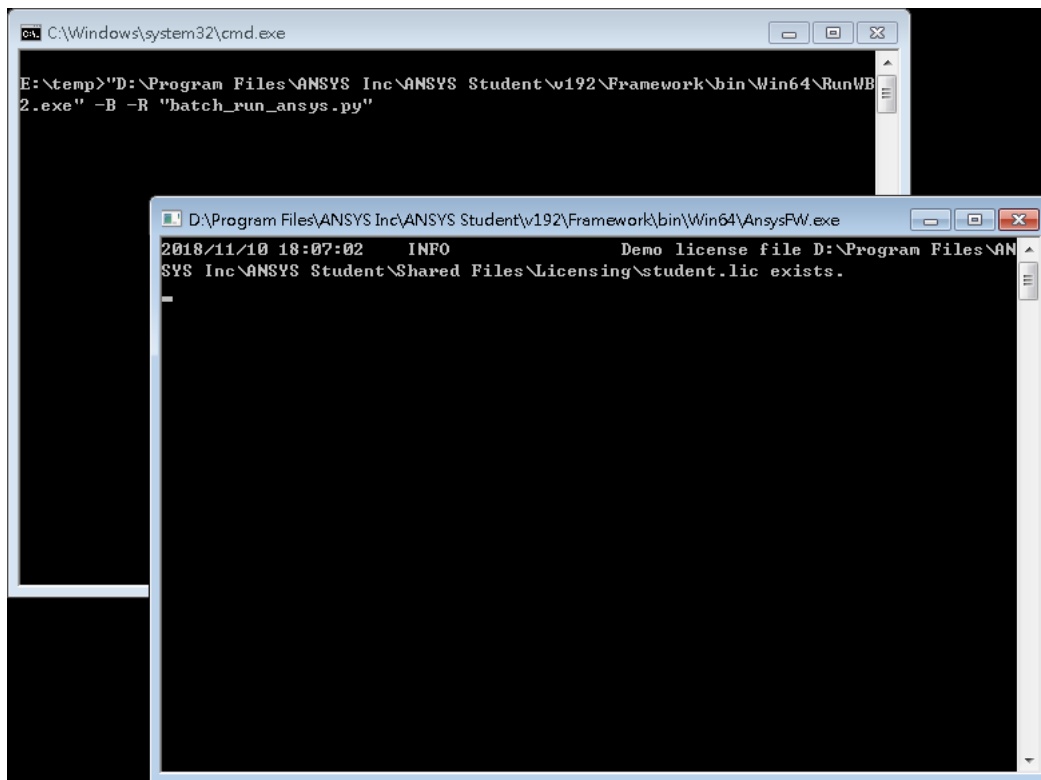


Figure 5: Execute batch_cmd.bat

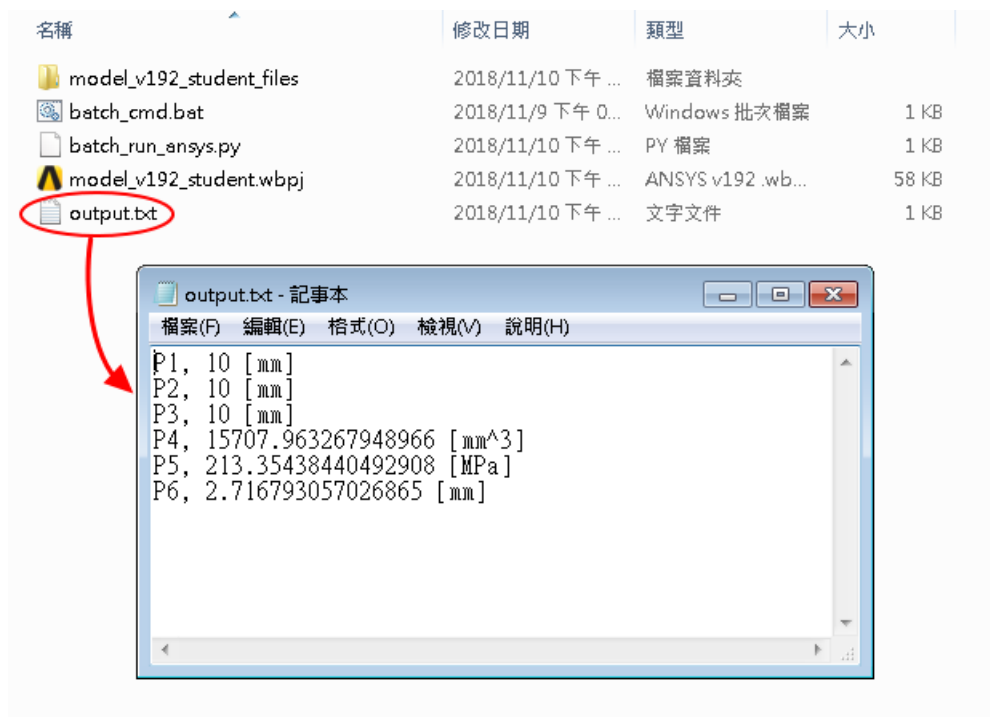


Figure 6: Output text file