

## 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), then execute the simulation and export results to a text file.

## 2 Install ANSYS Student

We are not encourage students to use commercial softwares without license. The ANSYS Student Software provide a twelve-month renewable license which is 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>

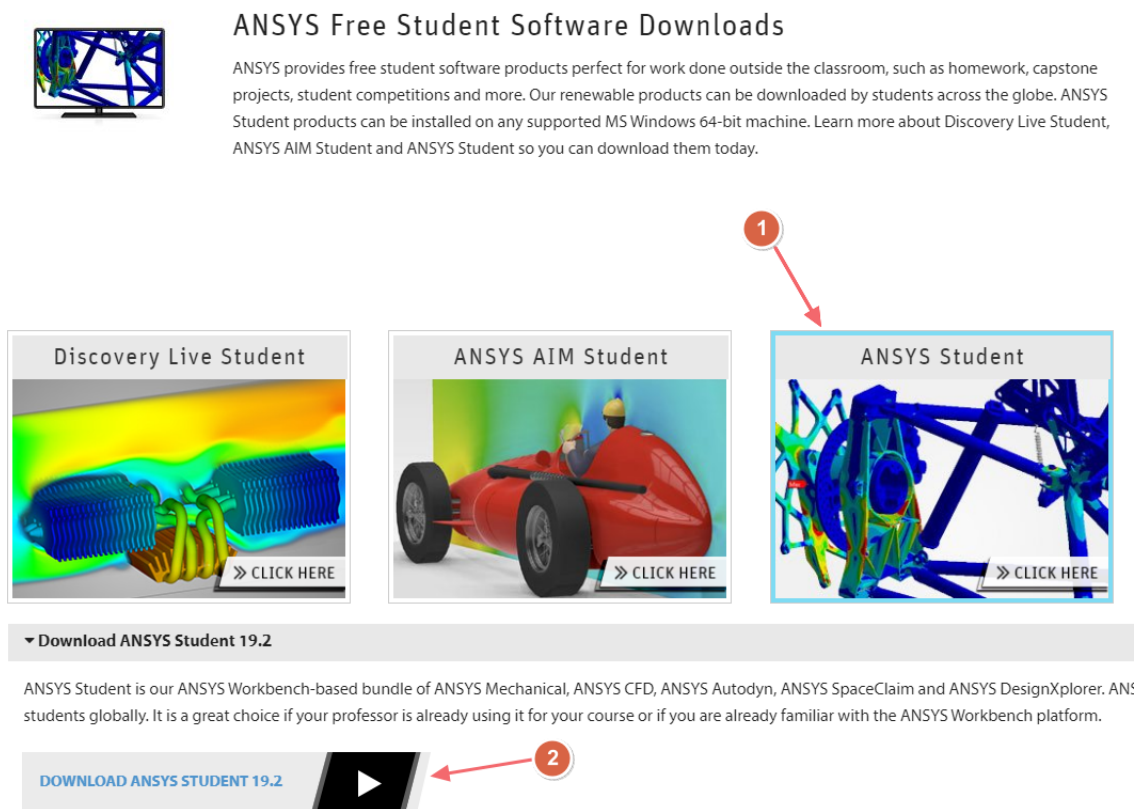


Figure 1: ANSYS Free Student Software Downloads

## 3 Example

### 3.1 Files

The **example.zip** should contain at least four 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 as the design variables.

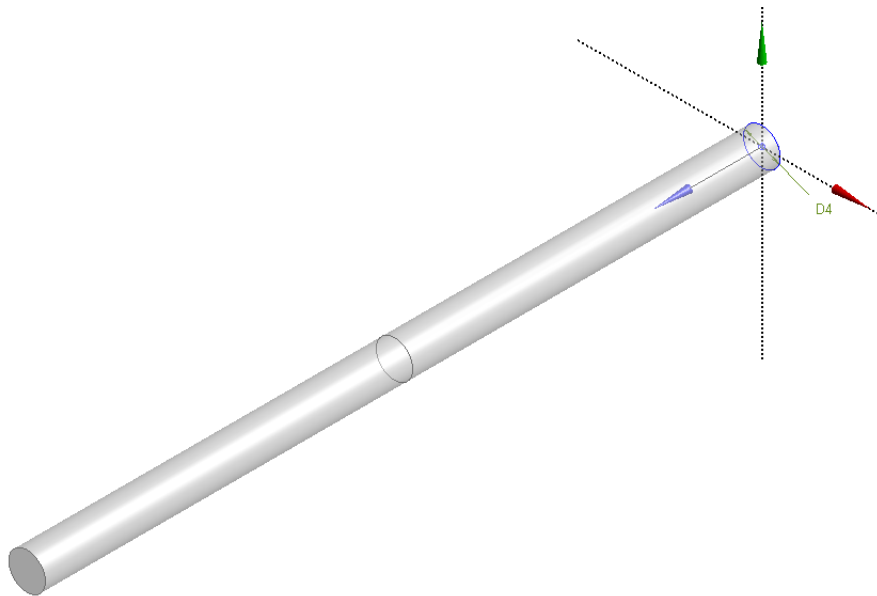


Figure 2: Geometry

In structural simulation, we fixed one end of the beam and apply a 100 [N] force (+x) on the other side. 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

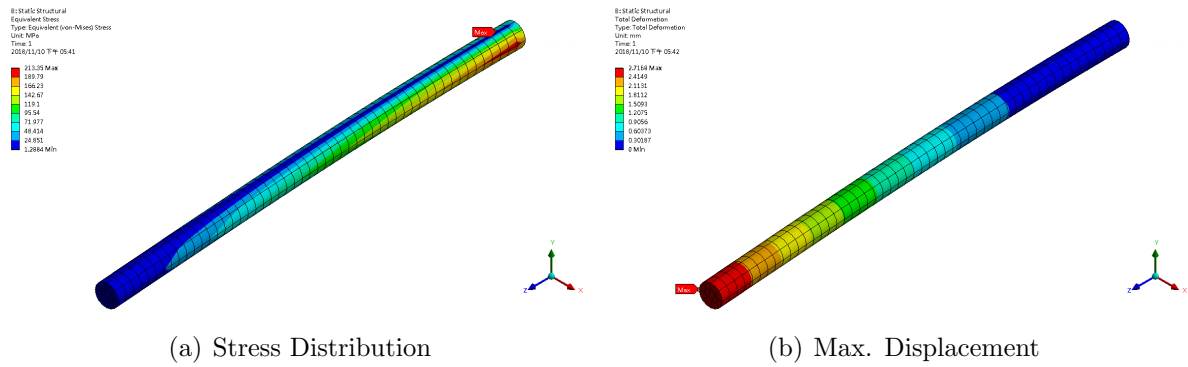


Figure 3: Analysis Result

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 <sup>3</sup>
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

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

You need to modify the project file's location in code line 3 before running the batch mode. This file path might be different in your computer.

```
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")
```

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

```
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()
```

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

- 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 executing the 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

---

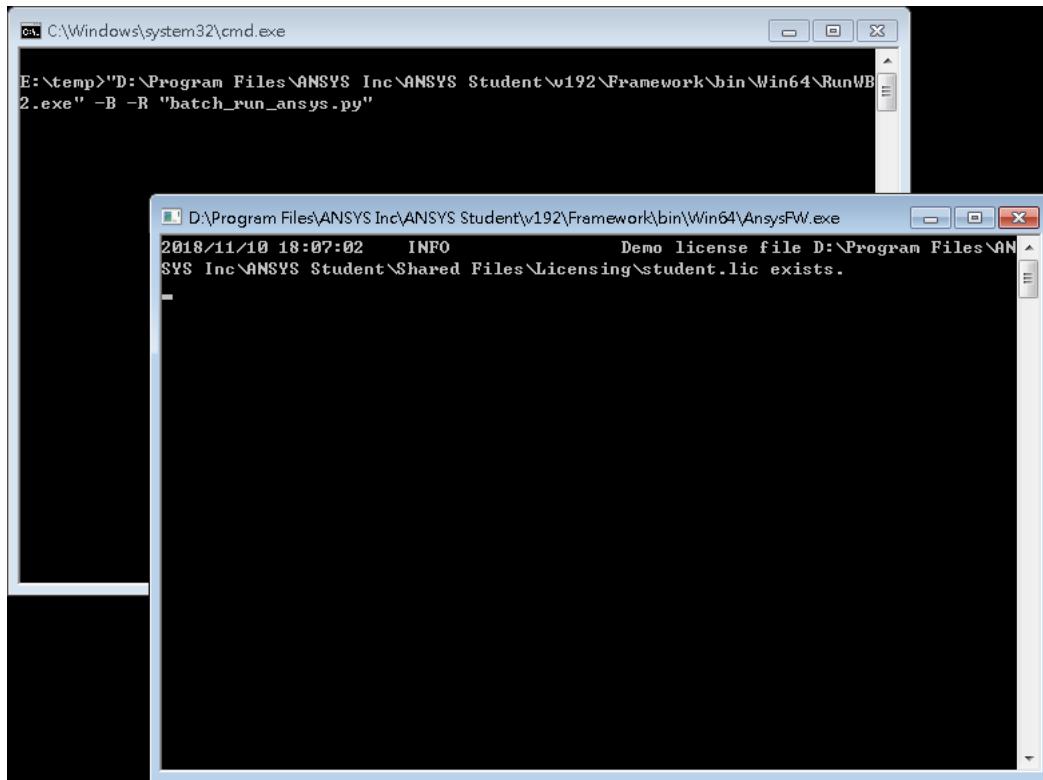


Figure 5: Execute batch\_cmd.bat

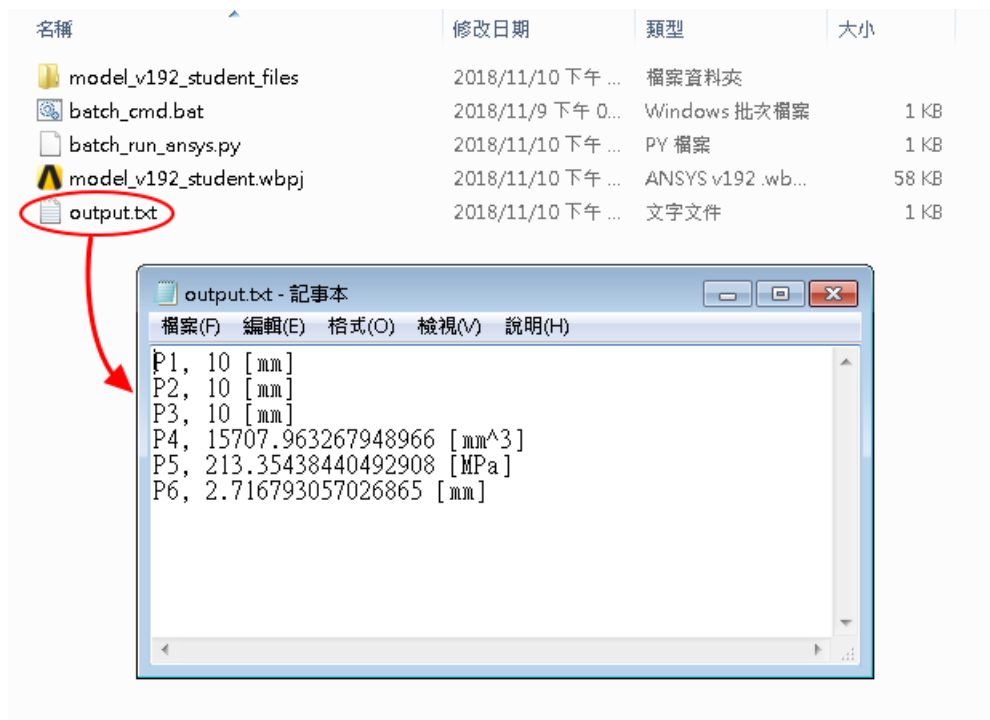


Figure 6: Output text file