1 Introduction

This document is only used for academic purpose and released in MIT License. We demonstrate how to change the ANSYS Workbench project's parameters (or variables) and execute the simulation in batch mode (command line) with Python script under Windows OS.

We are <u>**not**</u> encourage students to use commercial softwares. But the basic concept of the integration of your program with other softwares should be similiar.

2 Install ANSYS Student

Download the ANSYS Student Software from : **https://www.ansys.com/academic/free-student-products** and follow the installation instruction in the website.
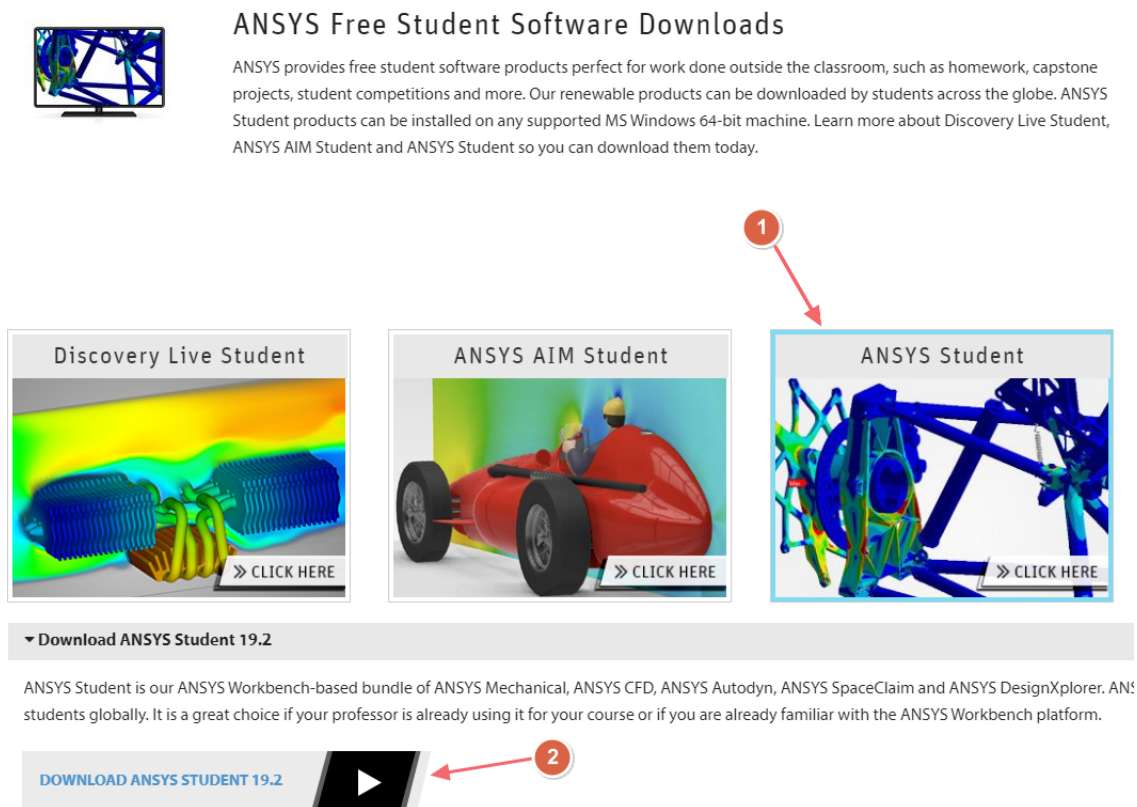

Figure 1: ANSYS Free Student Software Downloads

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.

3 Example

3.1 Download

Download the example file from 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 project file, 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.
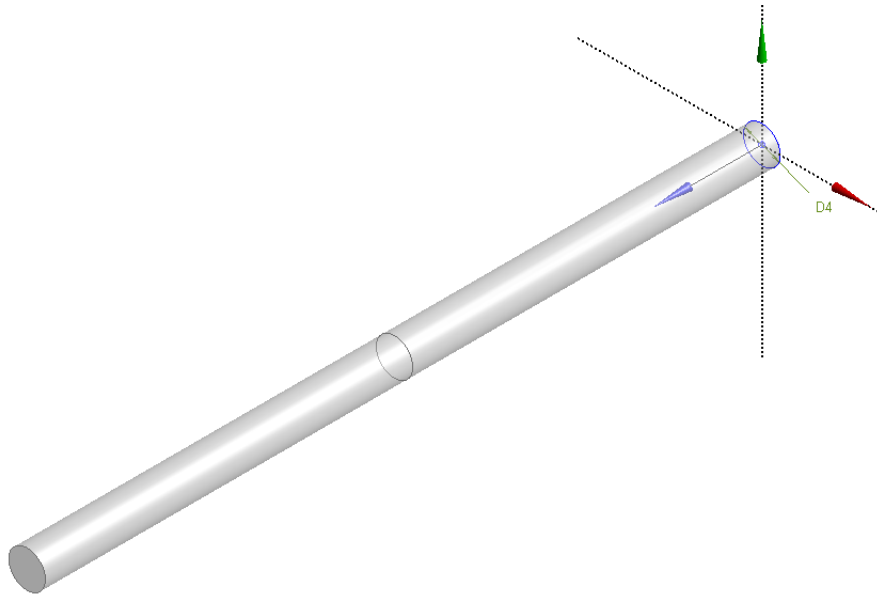

Figure 2: Geometry

In 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.

(a) Stress Distribution

(b) Max. Displacement
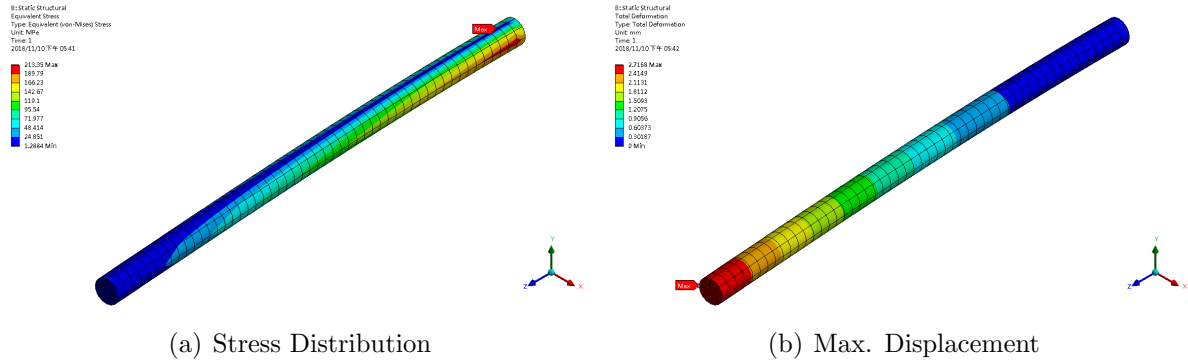
Figure 3: Analysis Result

Figure 4: Parameters

3.3 Batch Mode

- batch_run_ansys.py

  Before start running batch mode. We need to ckeck the project file 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 we 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 others mean running the program with batch_run_ansys.py in batch mode. This 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 :
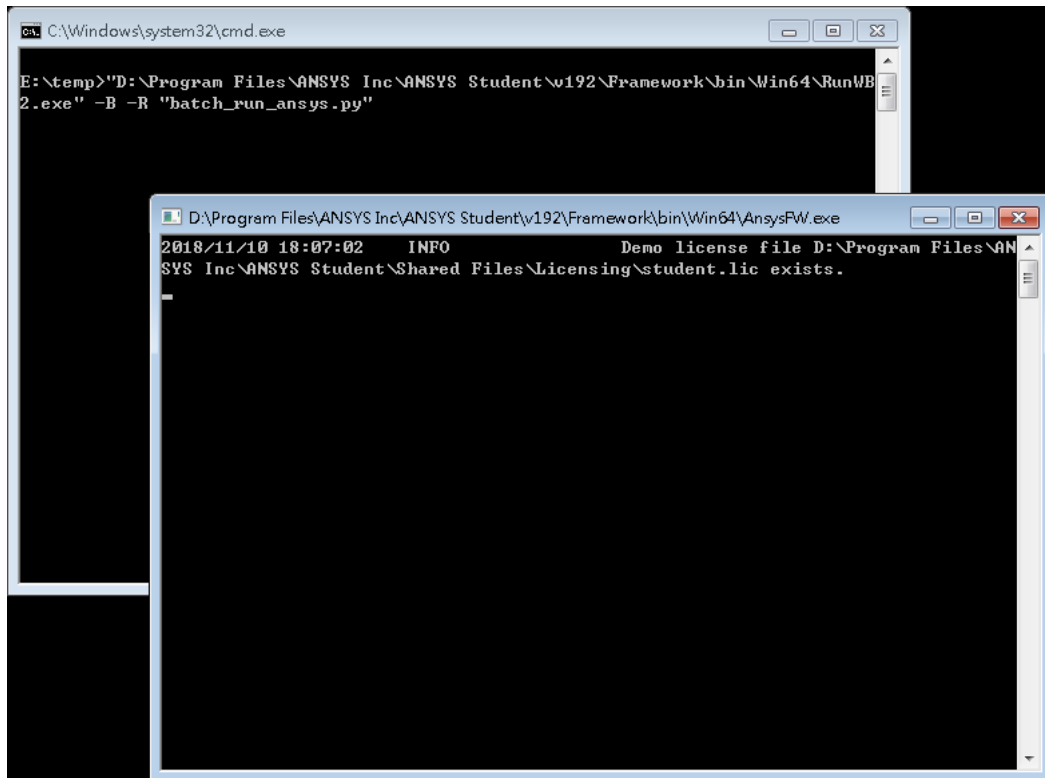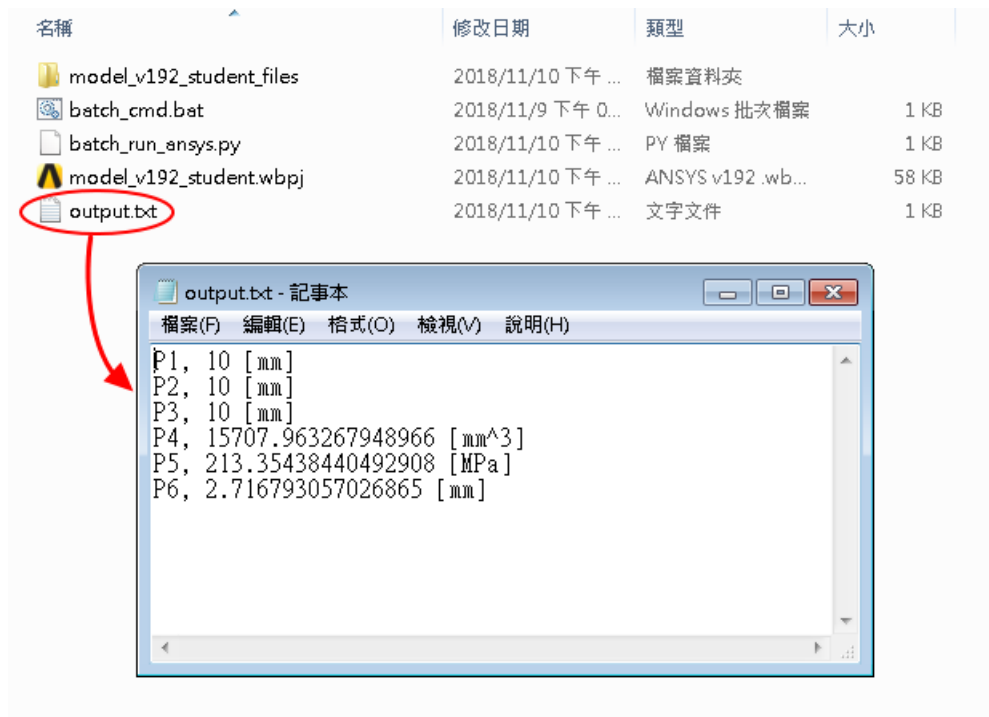**https://www.youtube.com/watch?v=jGWYmR0uqtU**

Figure 5: Execute batch_cmd.bat

Figure 6: Output text file