

INTRODUCTION TO *MultiSmart3D*

TUTORIAL

Based on the theoretical work in our group at the University of Akron, a software product was developed and a user-friendly GUI has been designed by incorporating the core code in FORTRAN and C++. The program is called *MultiSmart3D* which presents a forward calculation analysis of layered system. The tutorial for this program is described below.

MultiSmart3D program consists of an *exe* file. The program is connected with Fortran using the *lib* library file format. The program can be executed by double clicking on the *MS3D.exe* file.

Figure 1 shows the main window of the *MultiSmart3D* program. There are five tabs at the menu bar on the top of the window. The first one is **File** where the user can open an input file, print out, or do other common commands like any other Windows-based software. The second tab **Help** introduces the version of the program and its copyright information. The third one is **Input** where input information for the calculation can be set. **Calculation** and **Output** tabs are also designed for execution of the program and output representation respectively. If the **Input** tab is clicked, a drop-down listbox will be displayed (Fig. 2). By clicking the **Input check** button, a new window titled **General Info** will appear (Fig. 3).

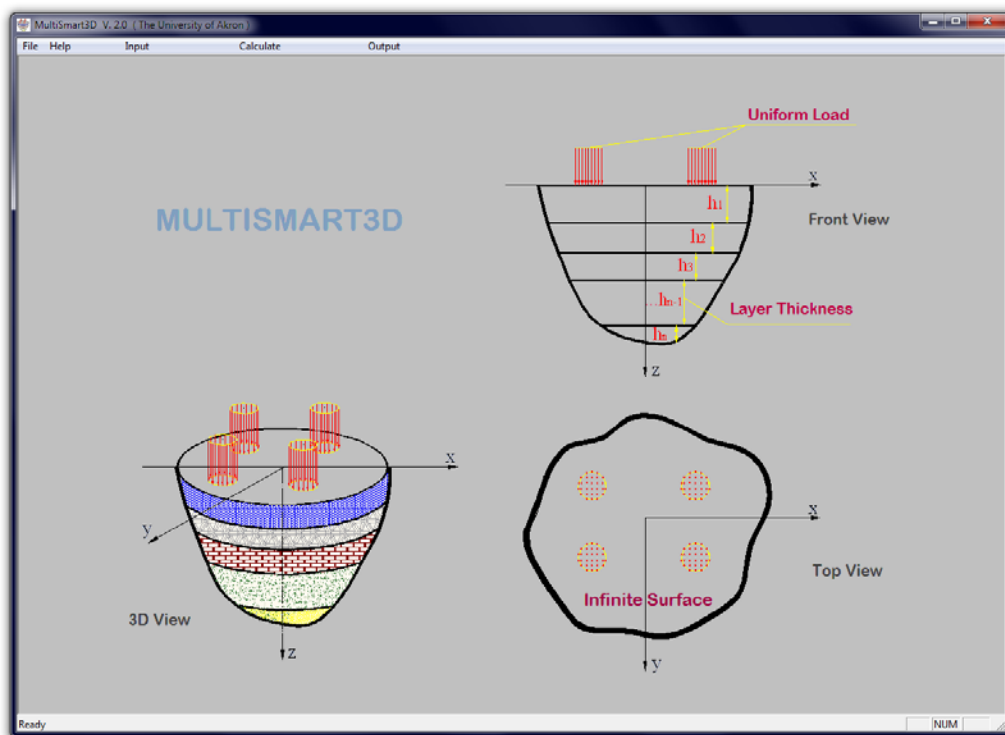


Figure 1 The main window of the GUI in *MultiSmart3D*.

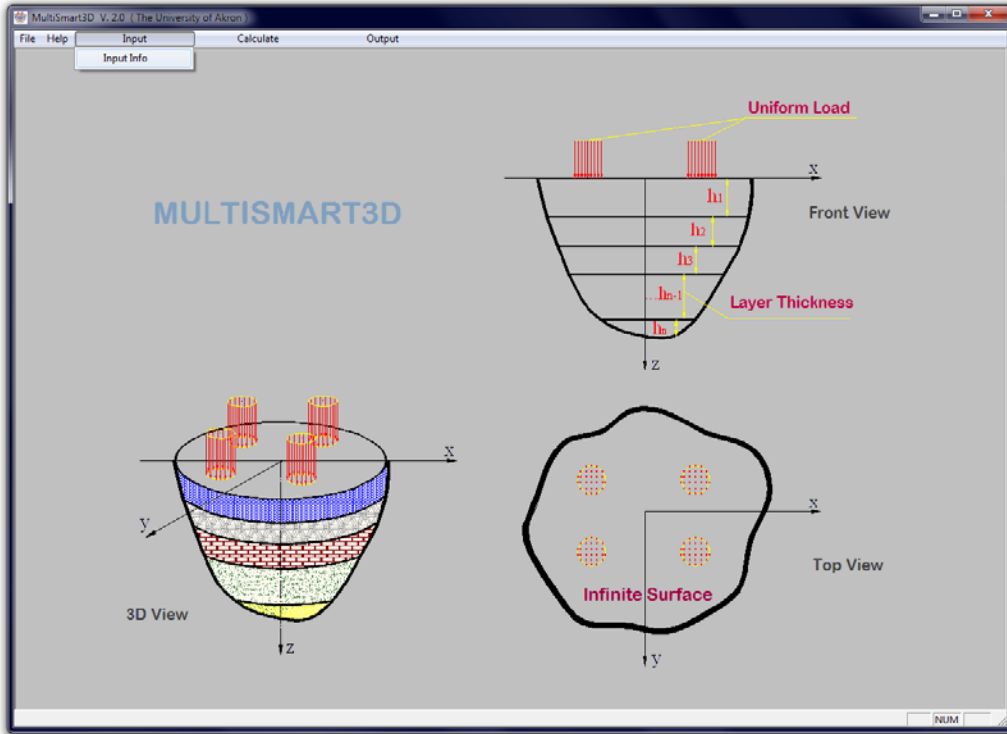


Figure 2 “Input” tab in *MultiSmart3D*.

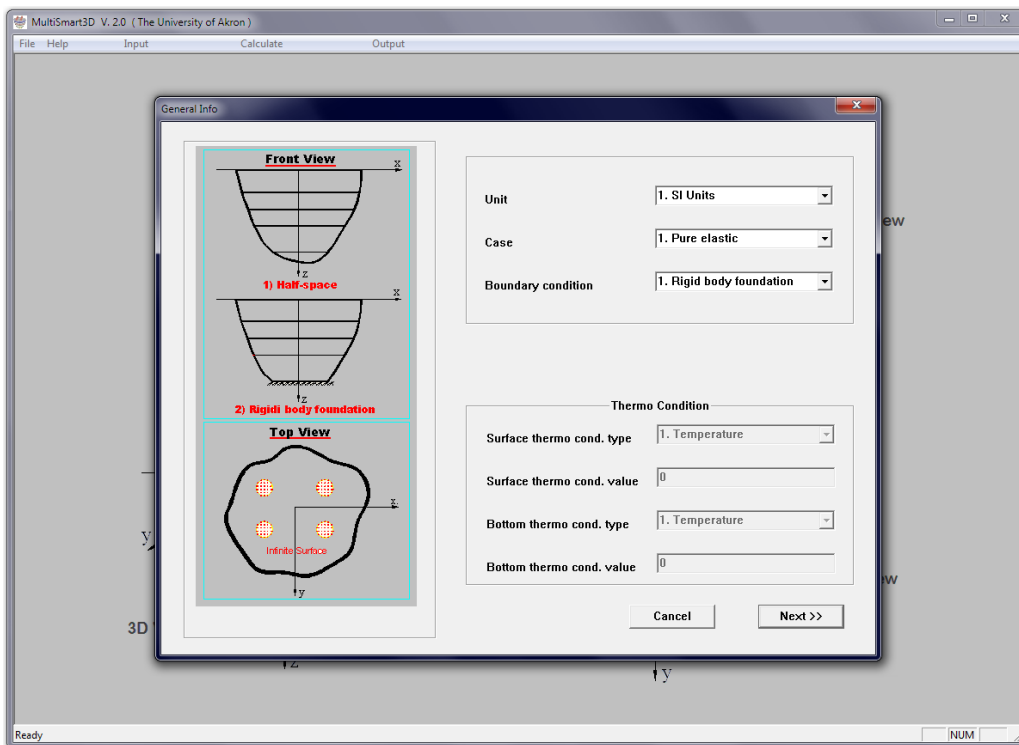


Figure 3 General information in *MultiSmart3D*.

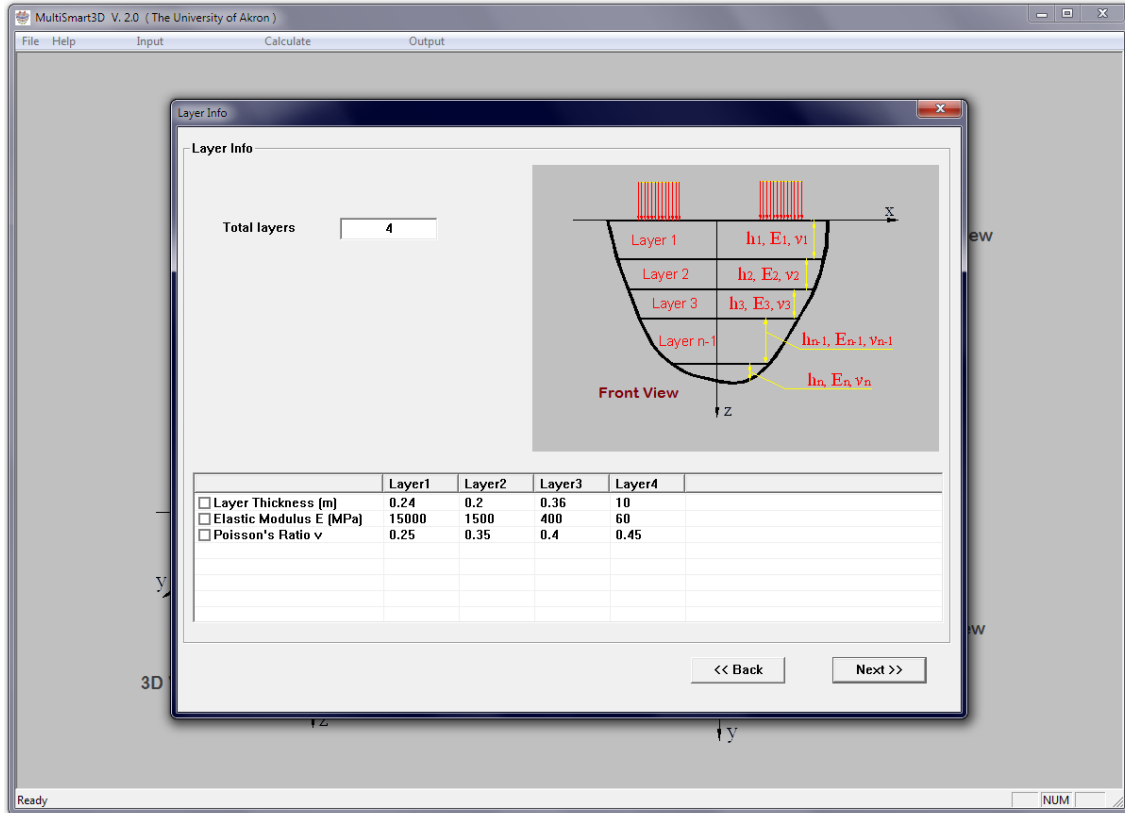


Figure 4 Layer information in *MultiSmart3D*.

In **General Info** dialog, different conditions can be set for the program including unit selection, case selection, and the type of boundary conditions. There are two separate sections in this window: **Initial Info** and **Thermo Condition**. In the **Initial Info** section the type of the units are to be set. The SI system of units is considered as default. Two options are presented here for the case of calculation: **Pure elastic** and **Thermo elastic**. The **boundary condition** can also be set to **Rigid foundation** or **Half space**. The current version of the *MultiSmart3D* program does not consider the US units and the thermo elastic boundary condition, which will be available in the future versions of the software.

After setting the general information in the first dialog of the GUI, we can continue by clicking the next button and a new window will appear (Fig. 4). This window is called **Layer Info** dialog and all data about pavement layers can be defined here. There are two ways to introduce this information to the program. The data can be read from a fixed format file or they can be imported manually. In the beta version of the program, data can only be imported manually. **Remark:** For the **halfspace** option, the thickness of the last layer can be any dummy number since the program will not need this value in the calculation.

The next dialog which will appear by clicking the **next** button is called **Load Info**. In this window the information about the number of loads, radius of applied loads, and their locations will be shown and the user can set them manually. This window is shown in Fig. 5.

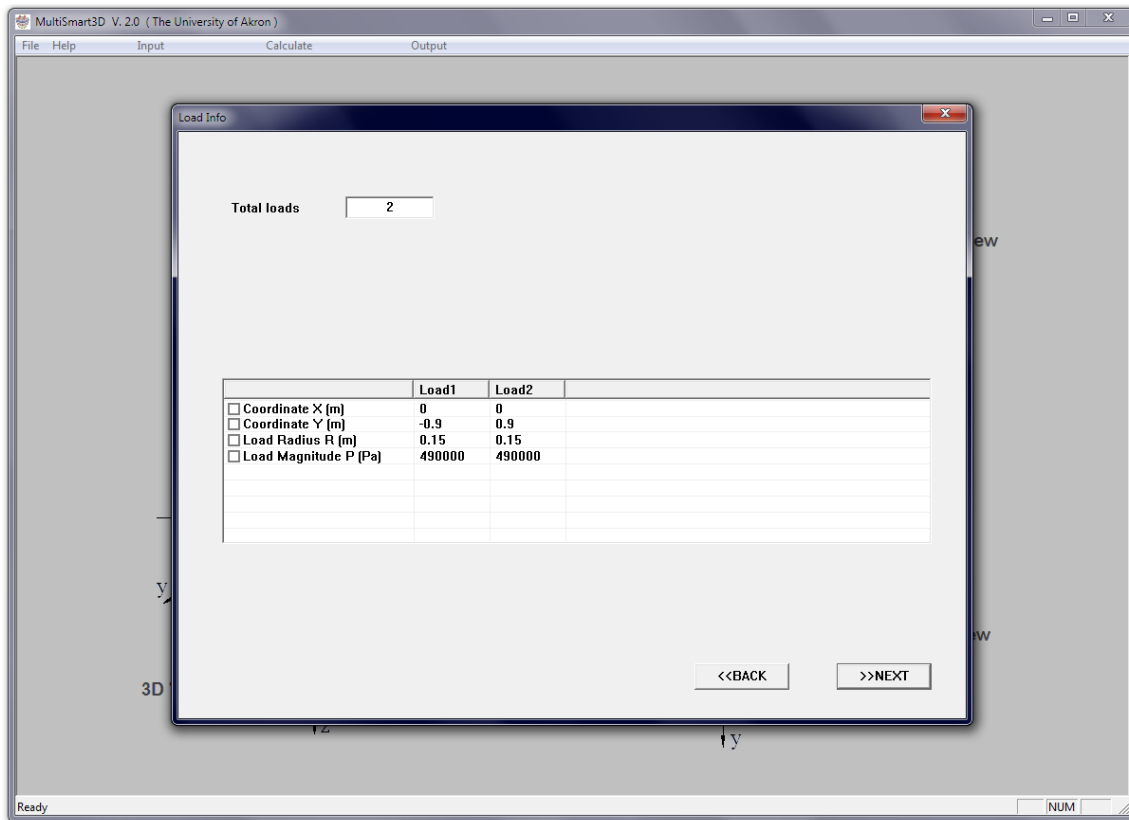


Figure 5 Load information in *MultiSmart3D*.

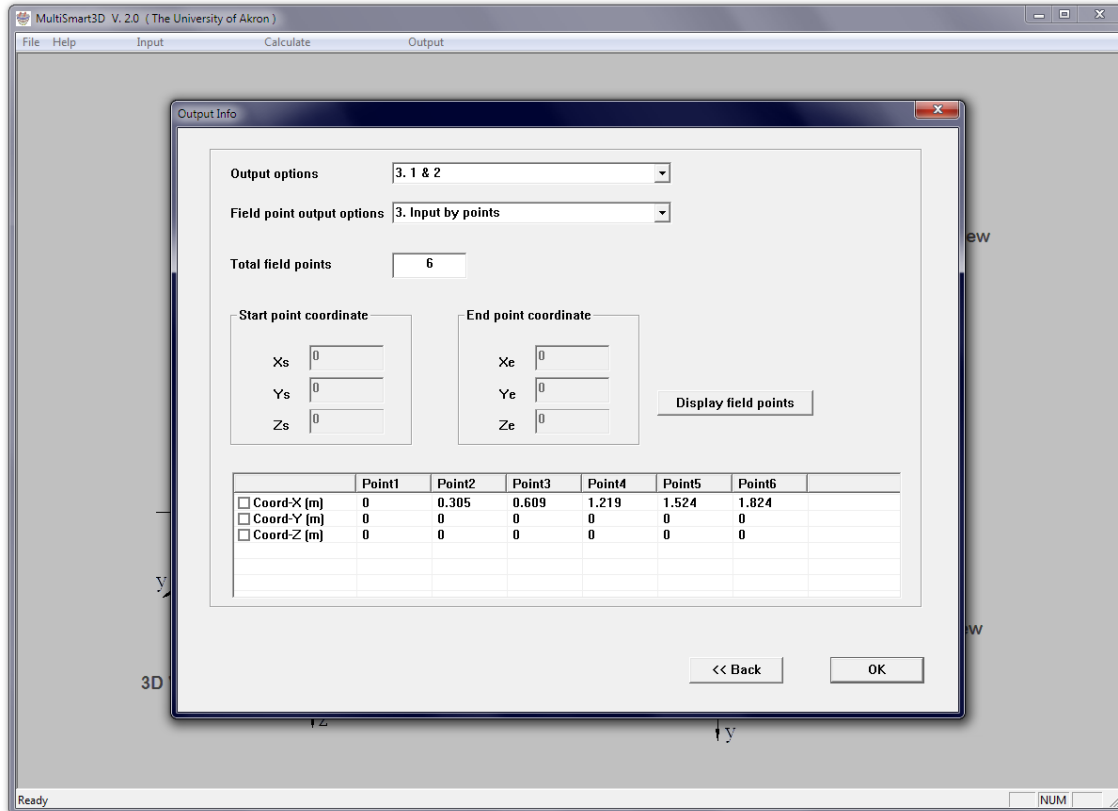


Figure 6 Output information in *MultiSmart3D*.

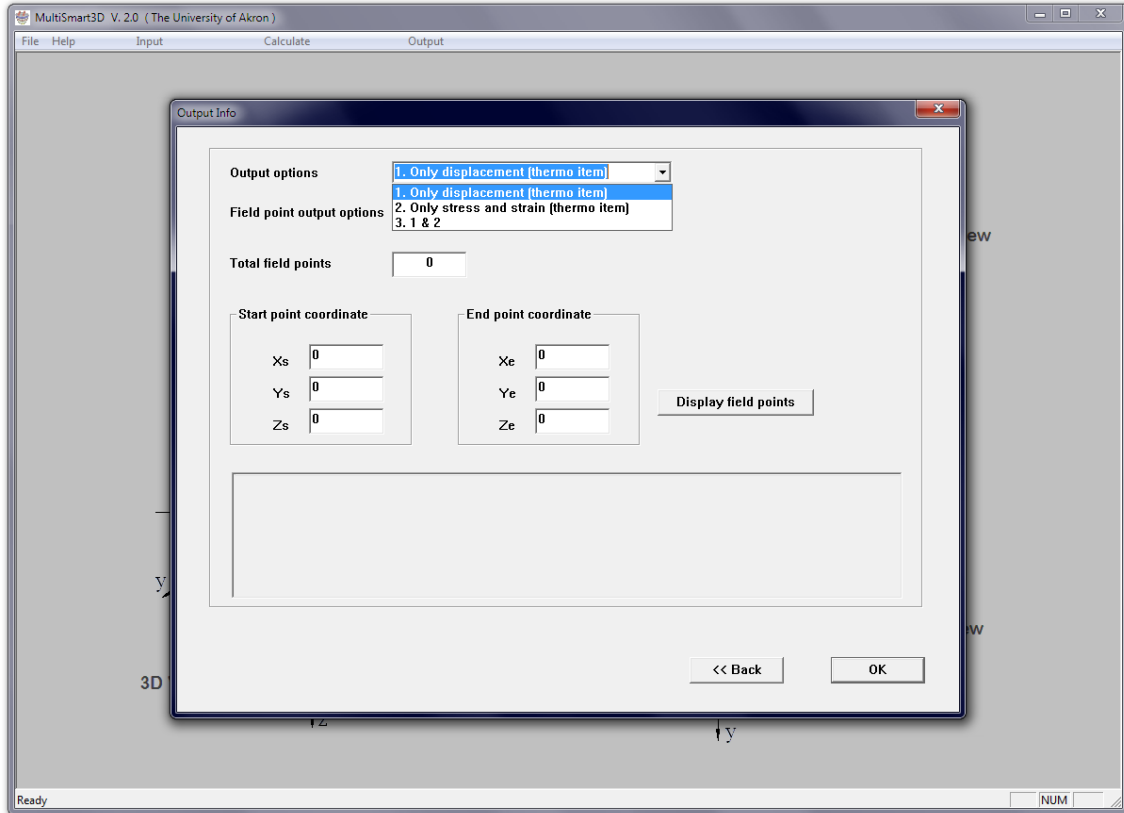


Figure 7 Output option for analysis in *MultiSmart3D*.

The last window is the **Output Info** dialog (Fig. 6) which is designed to determine the type of desired output by the user. The first option in this dialog allows the user to check the input data before execution and to select the desired output field (displacement, stress, and strain) (Fig. 7). The second option determines the position of the required field points by the user (Fig. 8). After hitting the **OK** button, the main window of the *MultiSmart3D* program will appear again. The program can be executed by clicking on the **RUN** in the **Calculate** tab of the menu bar (Fig. 9). A progress bar shows the improvement of the calculations (Fig. 10). The program starts to calculate the requested fields at required positions in the halfspace. A new window will pop up which states the end of the calculation procedure (Fig. 11). The user can access the results by clicking on the **Output** tab in the menu bar (Fig. 12). An example of the input check and output files are shown in Figs. 13-16.

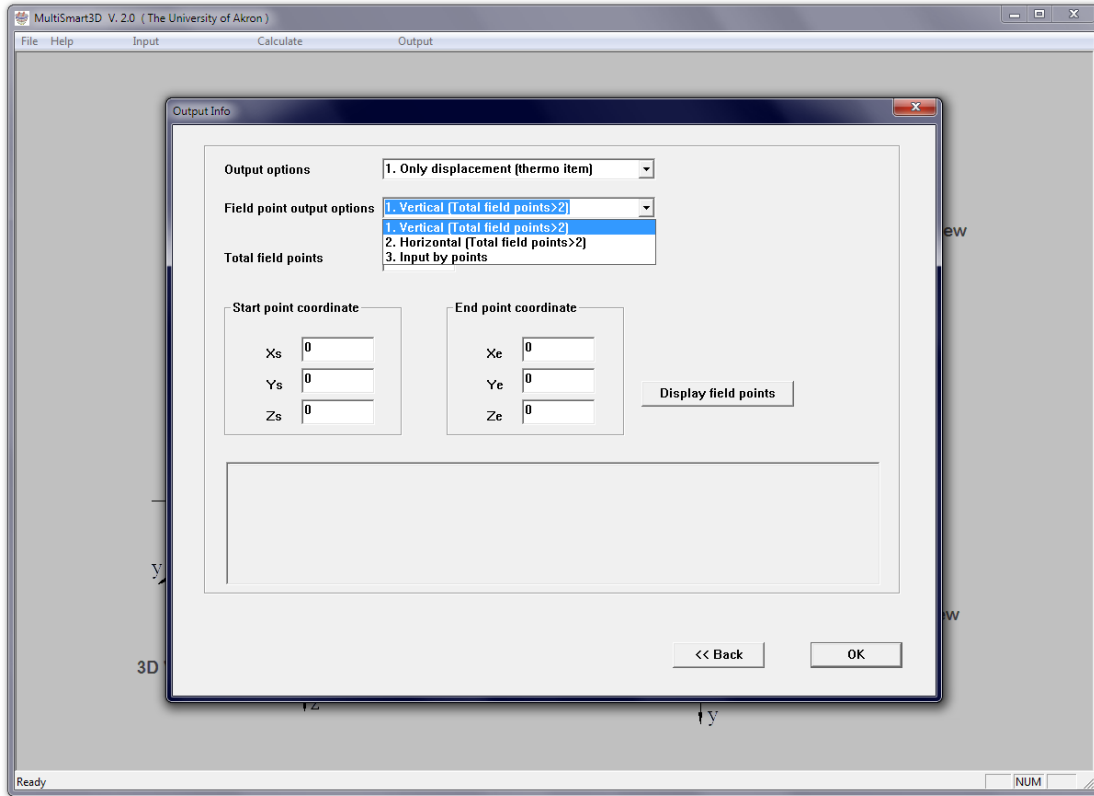


Figure 8 Field point option for analysis in *MultiSmart3D*.

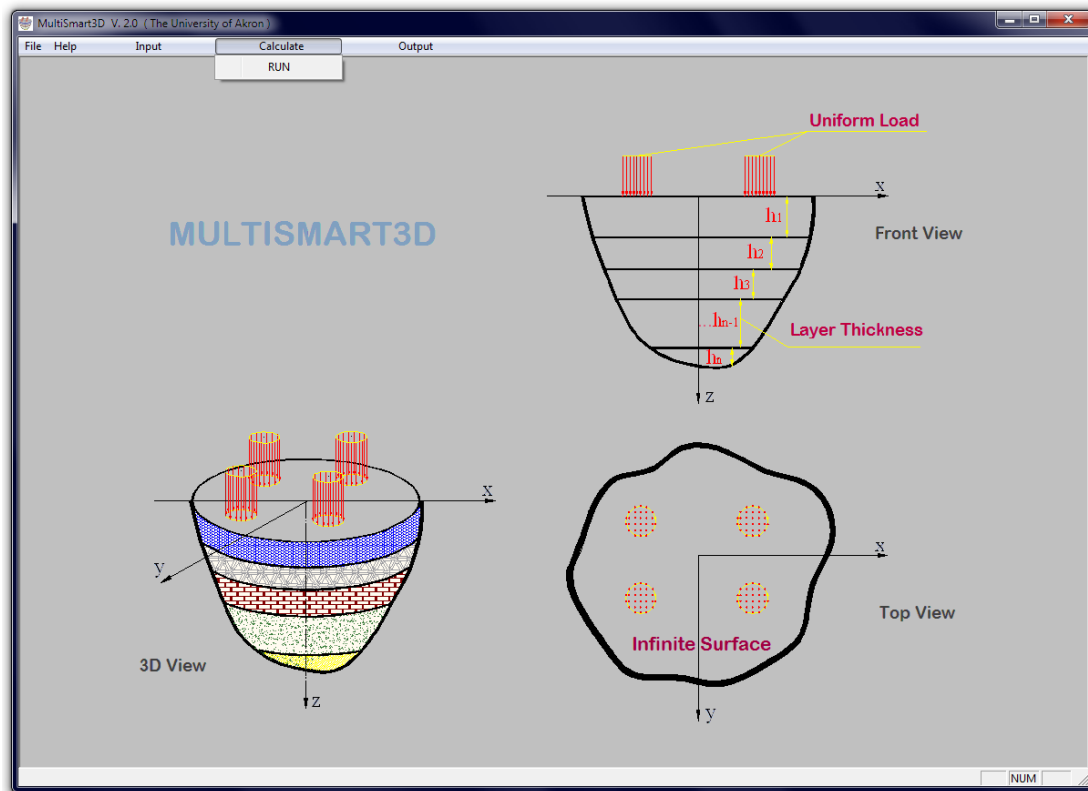


Figure 9 “Calculate” tab in *MultiSmart3D*.

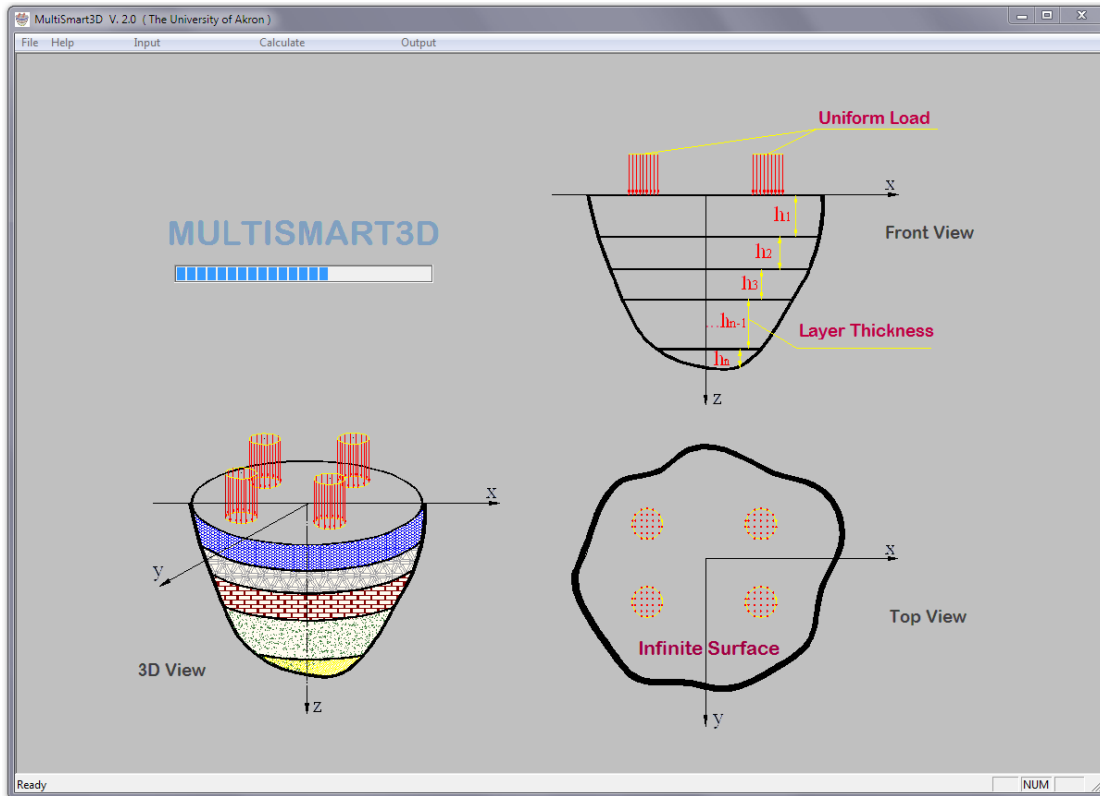


Figure 10 Progress bar in *MultiSmart3D*.

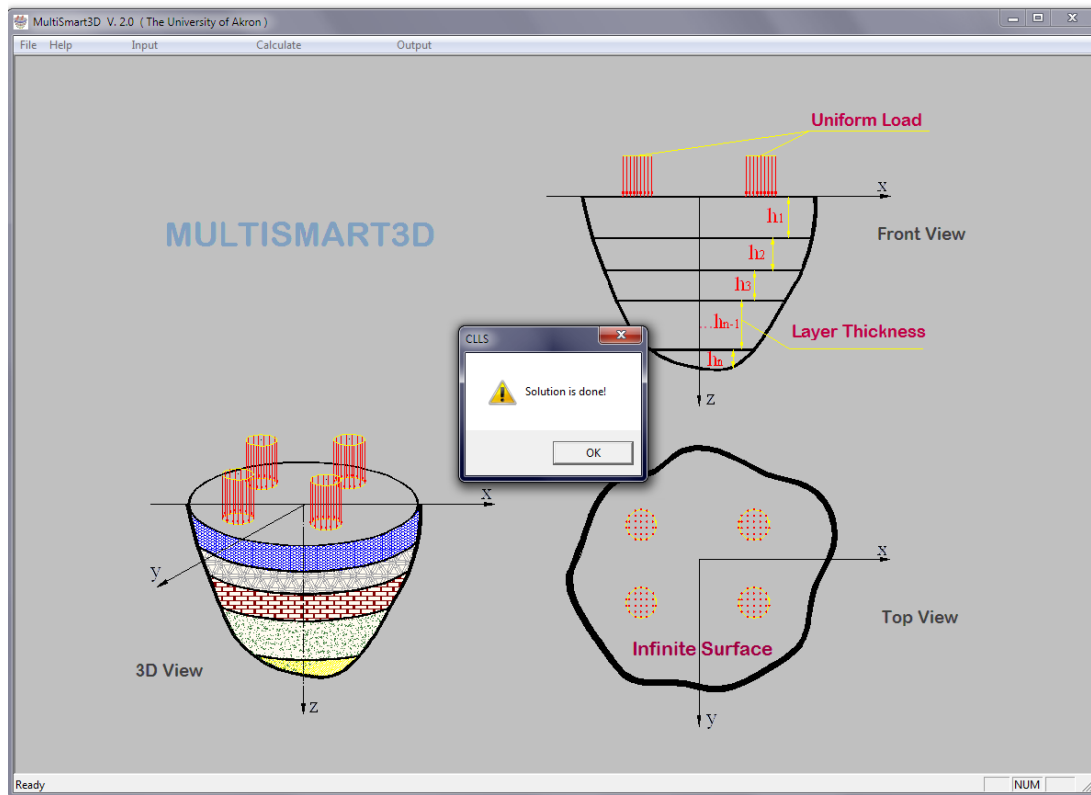


Figure 11 End of calculations in *MultiSmart3D*.

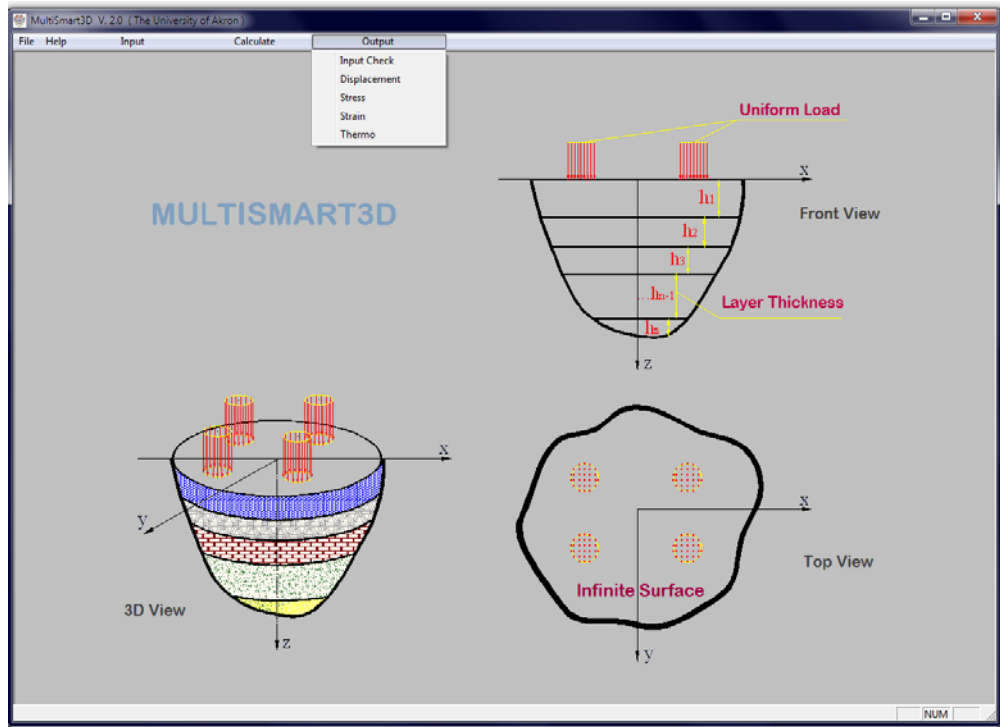


Figure 12 “Output” tab in *MultiSmart3D*.

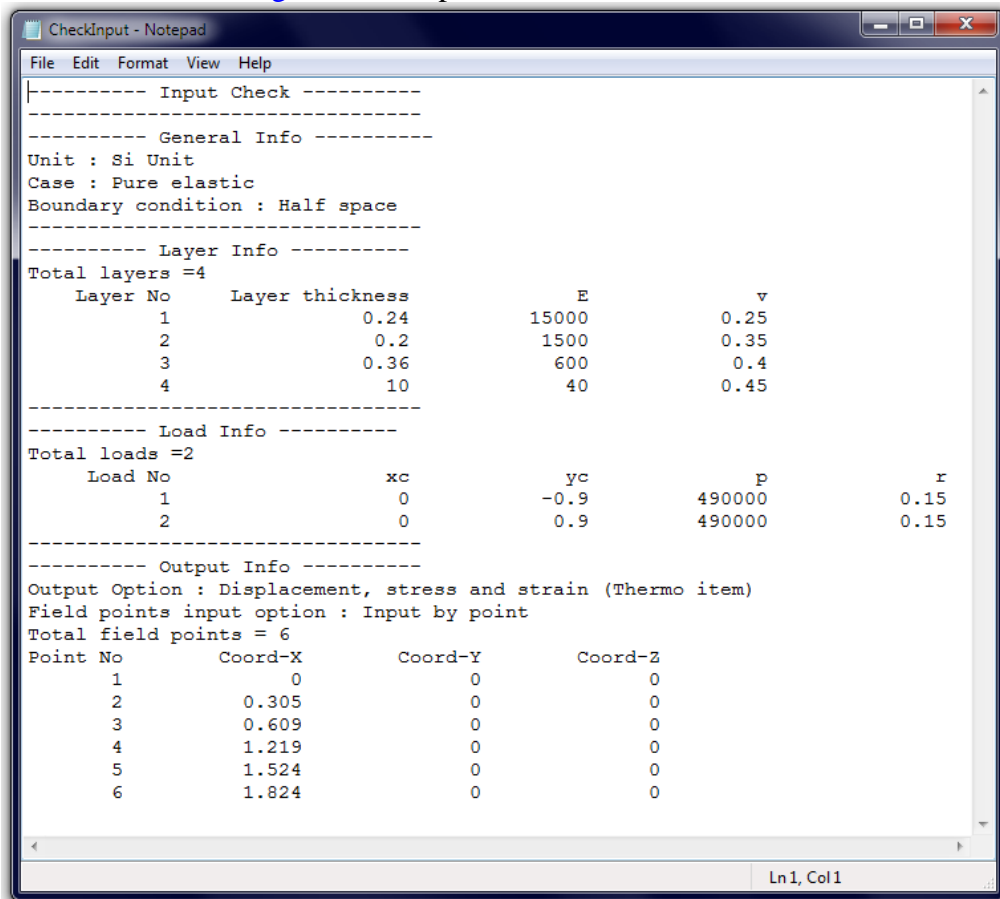


Figure 13 Input Check file in *MultiSmart3D*.

Displacement - Notepad

DISPLACEMENT of field points Unit: micrometer

No	Coord-X	Coord-Y	Coord-Z	Ux	Uy	Uz
1	0	0	0	7.46626e-016	0	303.395
2	0.305	0	0	-3.90809	-8.88178e-016	299.232
3	0.609	0	0	-6.76842	0	288.148
4	1.219	0	0	-9.19876	-4.44089e-016	255.363
5	1.524	0	0	-9.39064	-4.44089e-016	237.395
6	1.824	0	0	-9.2322	8.88178e-016	219.979

Ln1, Col1

Figure 14 Output Results file for displacement in MultiSmart3D.

Stress - Notepad

STRESS of field points Unit: Pascal

No	Coord-X	Coord-Y	Coord-Z	Sxx	Syy	Szz	Syz	Sxz	Sxy
1	0	0	0	-216217	-50658.6	0.000105312	0	0	2.0275e-011
2	0.305	0	0	-187059	-60364.7	8.24406e-005	0	0	-1.2008e-011
3	0.609	0	0	-128580	-74548.2	4.20232e-005	0	0	-4.00267e-012
4	1.219	0	0	-39647.3	-73138.5	4.42421e-005	0	0	4.00267e-012
5	1.524	0	0	-15209.2	-63843.4	-3.14374e-005	0	0	0
6	1.824	0	0	597.15	-53967.5	2.14588e-005	0	0	-8.00534e-012

Ln1, Col1

Figure 15 Output Results file for stress in MultiSmart3D.

Strain - Notepad

STRAIN of field points

No	Coord-X	Coord-Y	Coord-Z	SNxx	SNyy	SNzz	SNyz	SNxz	SNxy
1	0	0	0	-13.572	0.224021	4.45586	0	0	3.37903e-015
2	0.305	0	0	-11.4662	-0.909187	4.1312	0	0	-2.66454e-015
3	0.609	0	0	-7.35128	-2.82941	3.39187	0	0	-8.88178e-016
4	1.219	0	0	-1.42489	-4.21458	1.88392	0	0	0
5	1.524	0	0	0.0468887	-4.00361	1.32084	0	0	4.44089e-016
6	1.824	0	0	0.935686	-3.60815	0.89213	0	0	-1.33227e-015

Ln1, Col1

Figure 16 Output Results file for strain in MultiSmart3D.