BESO2D Manual – Getting started with BESO2D

By Z.H. Zuo

This part of the manual contains two sections: *Getting started with GUI usage of BESO2D* and *Getting started with Command Line usage of BESO2D*. BESO2D can be used in two individual ways: GUI usage and Command Lind usage. The user can choose either way and get a quick instruction for the corresponding way of using BESO2D in the following.

Getting started with GUI usage of BESO2D

This section demonstrates, step-by-step, how to use the BESO2D *graphic user interface* (GUI) window, from constructing the structural model to obtaining the final optimized topology. The procedure includes the following steps:

1. Drawing the design domain of a structure.
2. Generating a finite element mesh of the design domain.
3. Specifying boundary conditions, loading conditions and material properties.
4. Performing FEA on the meshed model and showing the analysis result.
5. Performing BESO optimization.
6. Viewing the final optimal design and the evolution histories.
7. Continuing optimization from an obtained design.

To illustrate these steps, a typical Michell type structure shown in Figure 1 is used as an example.

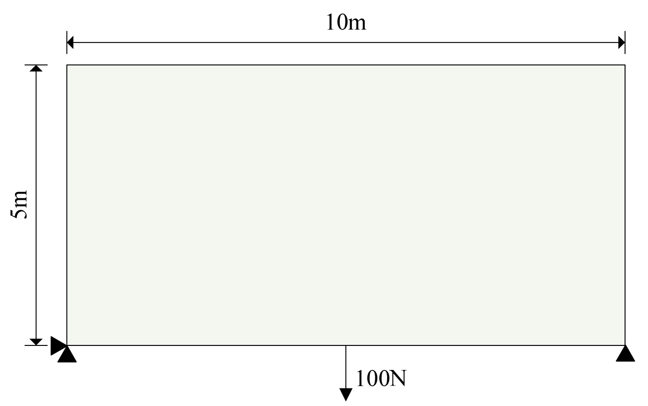


Figure 1 Design domain of a Michell type structure.

1. Drawing the design domain of a structure

This step comprises of the following actions:

- Start BESO2D.

- Set the screen width: click on the  button in the **Draw/Run Toolbar** and input 15.0 in the "Width of display area" dialogue and click on "OK".

- Draw a rectangle: click on the  button in the **Draw/Run Toolbar**, locate the mouse at the position of (1.0, 2.0) (see the current position of the mouse in the Status bar), press the left mouse button, drag the mouse to the position of (11.0, 7.0) and release the mouse. A rectangle is obtained in the display area as shown in Figure 2.

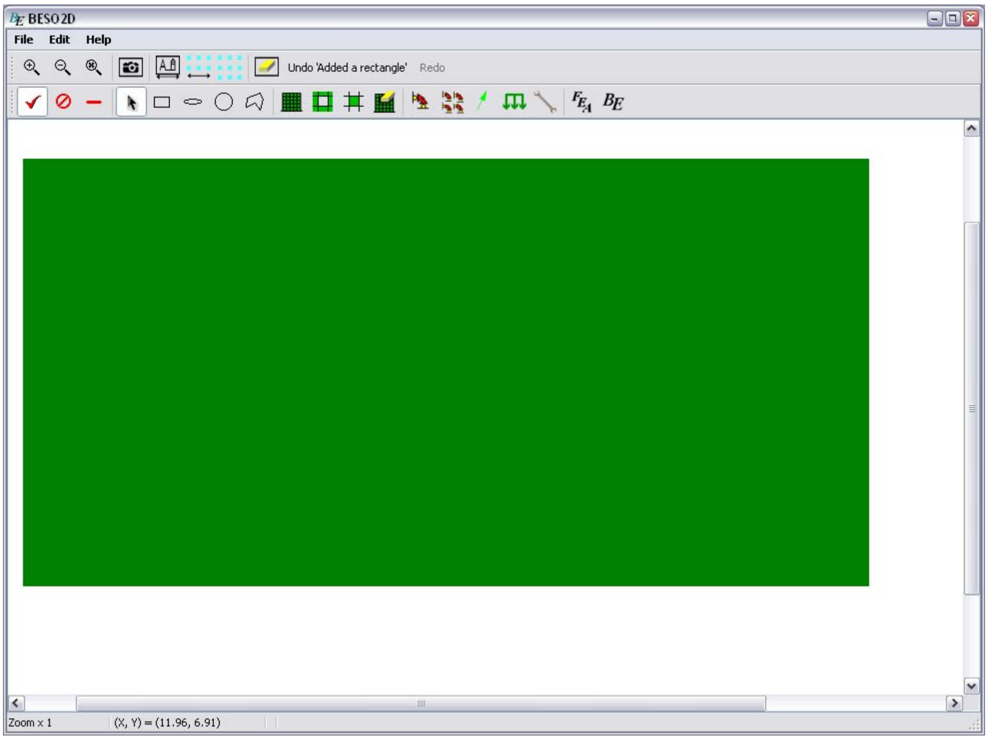
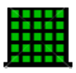


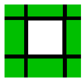
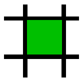
Figure 2 A rectangle is drawn in the BESO2D window.

2. Generating a finite element mesh of the design domain

After the geometry of the structure is defined, the model is ready to be meshed in this step.

- By clicking on the  button in the **Draw/Run Toolbar,** the "Generate mesh" dialogue will appear. Input 0.10 for the distance between two adjacent nodes and then click on "Generate". Now the meshed structural model is obtained as shown in Figure 3.

Note that when the mesh is generated, the corners of the structure may sometimes be shifted slightly to nearby locations and thus the geometry of the design domain is changed. This occurs when the length of the structure divided by the distance between two adjacent nodes does not yield an integer.

Therefore, it is recommended to check of the coordinates of the corner nodes after mesh generation. If necessary, the user may add or delete elements at the edges to maintain the intended dimensions for the structure. Deleting or adding elements can be done by clicking on  or  buttons and then click the mouse at the relevant location.

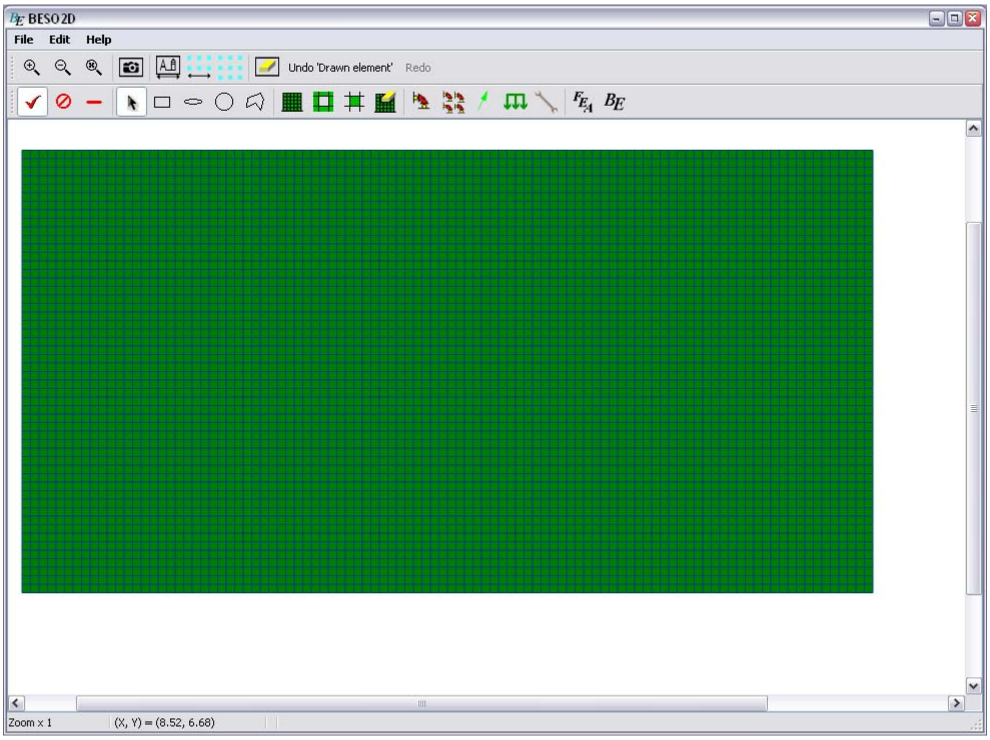
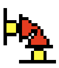
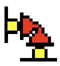


Figure 3 A finite element mesh is generated for the structural model.

3. Specifying boundary conditions, loading conditions and material properties

After the structural model is meshed, we can apply boundary and loading conditions and define the material properties.

- By clicking on the  button in the **Draw/Run Toolbar**, the "Constraint" dialogue will appear. Select "All degrees of freedom fixed" and click on "OK". Now the dialogue will disappear. Move the mouse close to the left bottom corner of the meshed rectangle and click the left mouse button to add the first constraint. By clicking on the  button again to open the "constraint" dialogue and select "Y-Degree of freedom fixed". Click "OK" and apply the second constraint to the right bottom corner of the rectangle as the first constraint. Note that one does not need to position the mouse exactly on the node but can click on a position which is near enough - the constraint will be automatically applied to the nearest node.

- By clicking on the  button in the **Draw/Run Toolbar**, the "Point load" dialogue will appear. Input the force components as 0 for Force-X and -100.0 for Force-Y. Click on "OK" and the dialogue will disappear. Move the mouse to the position of (6.0, 7.0) (the mouse position can be seen from the Status bar) and press the left mouse button to apply the force. The structural model with boundary and loading conditions is shown in Figure 4.

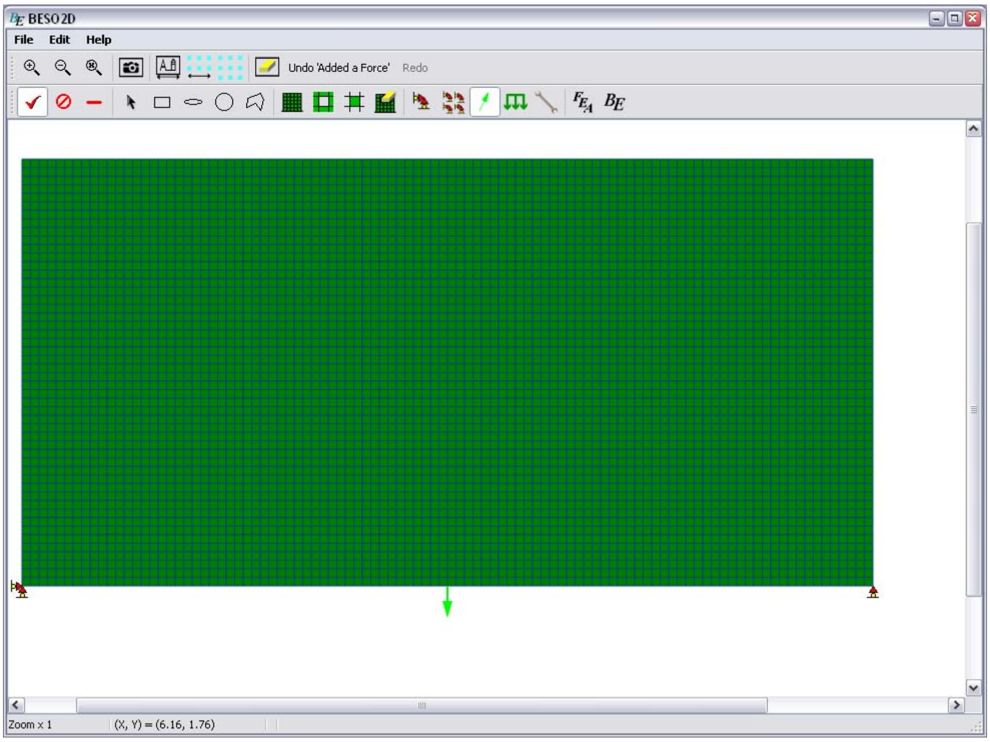
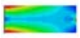


Figure 4 Structural model with boundary and loading conditions.

- By clicking on the  button in the **Draw/Run Toolbar**, the "Property" dialogue will appear. Enter the material properties such as 2.1E+11 for Young's modulus, 0.31 for Poisson's ratio and 7800.0 for mass density. Note that in current BESO2D the stiffness optimization is carried out without considering the gravity loading. Therefore, the mass density does not affect the optimal design.

4. Performing FEA on the meshed model and showing the analysis result

Now the model is ready for finite element analysis and BESO optimization. Although an FEA before optimization is not compulsory, it is always helpful and therefore strongly recommended to perform an FEA to check whether the model is constructed correctly by checking the FEA result, e.g. the von Mises stress distribution. The steps are explained below.

- Click the  button in the **Draw/Run Toolbar**. A message box to indicate that the FEA is running pops up. The computation time depends on the total number of elements and the computer hardware configurations.
- When the FEA is finished, the von Mises stress distribution will be automatically displayed on structural model. The stress levels can be found on the right hand side of the display area. Note that the **View Toolbar** now appears in the toolbar area.
- Chick on  and  buttons in the **View Toolbar** to switch on or off the deformation and stress distribution.
- The initial model for the Michell type structure is correctly constructed if the obtained FEA result is similar to that shown in Figure 5.

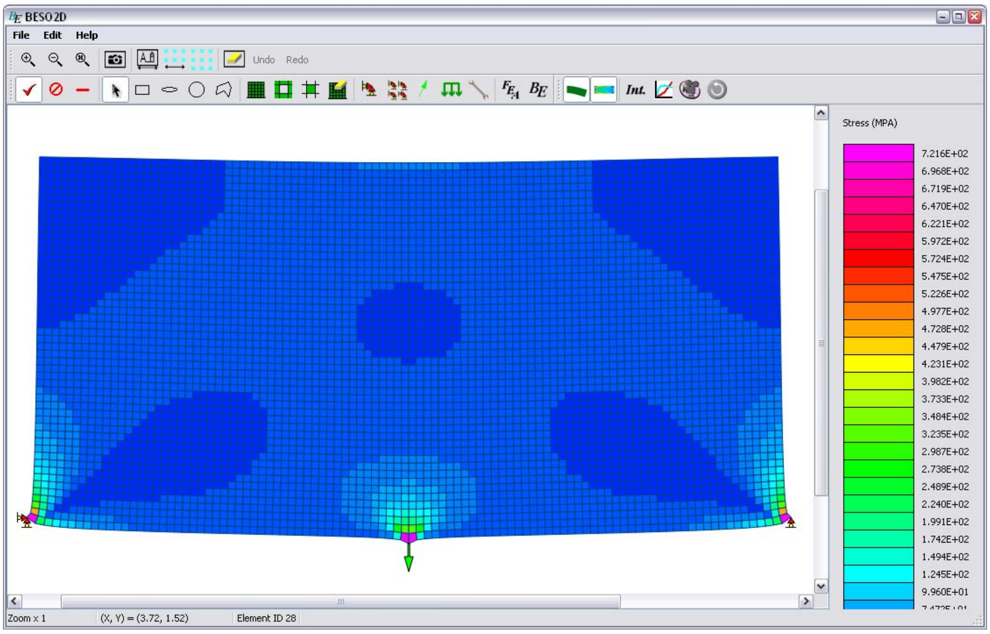


Figure 5 Stress and displacement distributions of the initial model.

5. Performing BESO optimization

To perform BESO on the structure, follow these steps.

- Click the  button in the **Draw/Run Toolbar.** Enter (or accept) the parameters in the "Start BESO" dialogue. The parameters include evolutionary volume ratio (*ER*), filter radius ($r_{\min}$), convergence tolerance τ, maximum iteration number and the objective volume fraction.
- The evolutionary volume ratio controls the volume fraction change between two consecutive iterations. In this example, set *ER* = 2.0%.

- To overcome checkerboard pattern and mesh-dependency problems, the filter radius is recommended to be 2~3 times the element size. In this example, set the filter radius to 0.20.
- The convergence tolerance gives an allowable convergence tolerance such as 0.10% in this example.
- The maximum iteration number is set to be 200 in this example.
- The objective volume fraction defines the final material usage as a percentage of the material in the design domain. Set 60.0% for this example.
- Make sure the “New optimization run” checkbox is checked. Now the “Start BESO” dialogue shown in Figure 6 will appear.

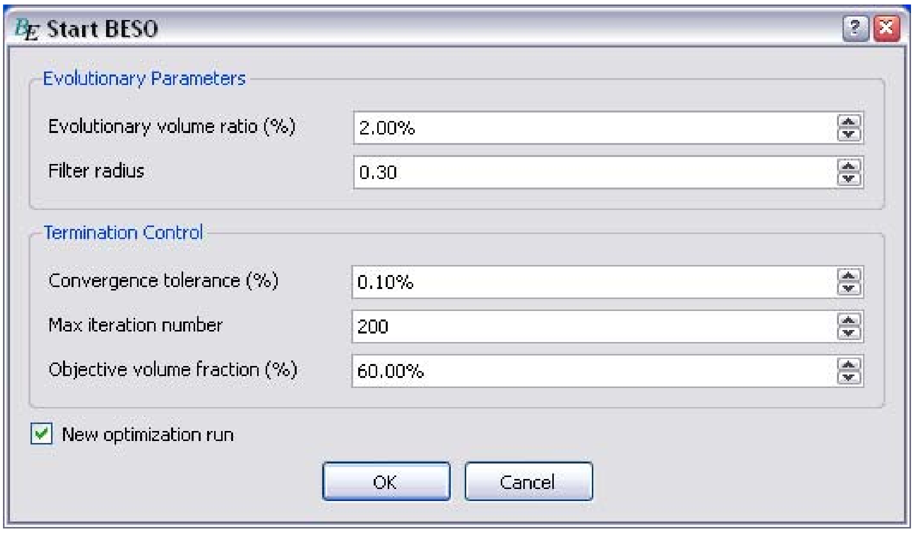
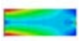


Figure 6 The BESO parameters used in this example.

- Now click on the “OK” button. The optimization starts immediately and a “BESO running” message pops up.
- After the pre-processing, the “BESO running” message box will display information indicating the current iteration number. During the optimization process, the intermediate designs are shown in the display area. Note that if the  button is pressed down, the real-time stress distribution will be displayed on the structure like Figure 7. The user can see how the structure is evolving on the screen.

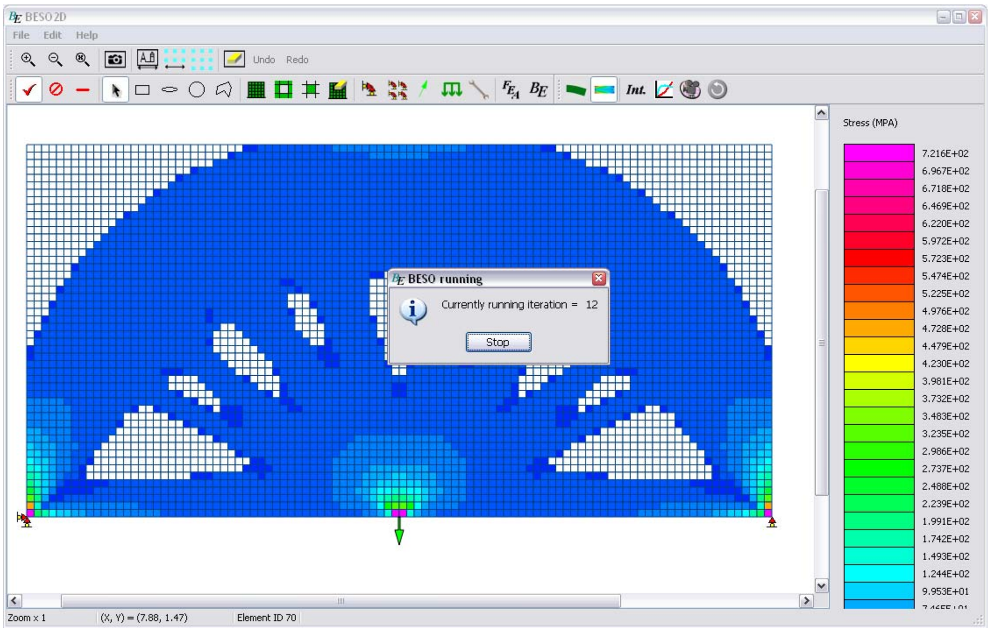


Figure 7 An intermediate design during the optimization process.

6. Viewing the final optimal design and the evolution histories

- After a short while, a "BESO finished" message box will show up indicating the end of the optimization run with the message "Optimization converged to a solution". Click on "OK" to dismiss the message and the final optimized design is shown in the display area as in Figure 8.

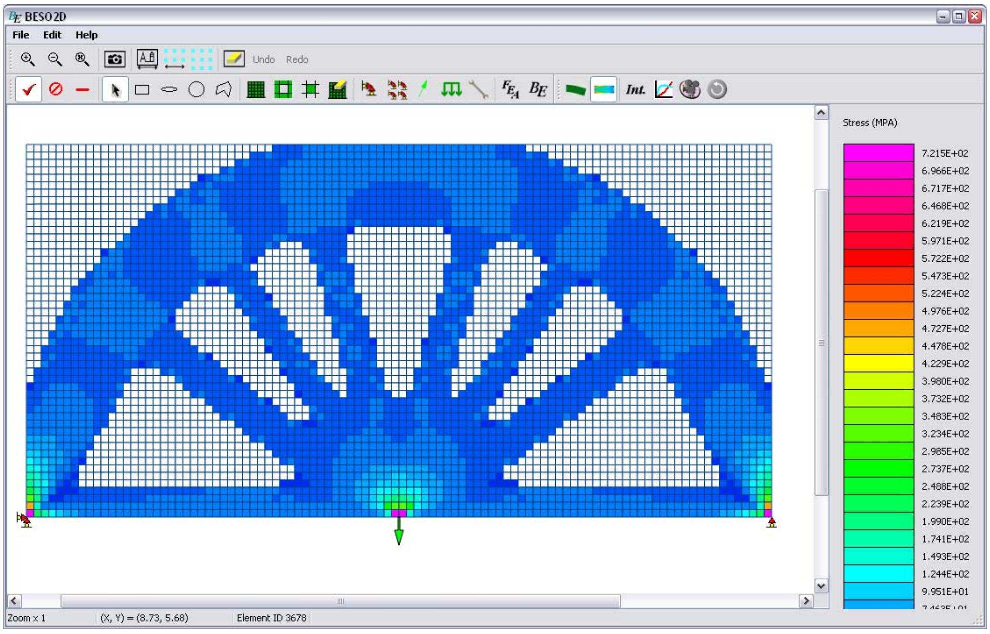
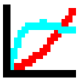


Figure 8 The final optimal design of the Michell type structure with 60% volume fraction.

- Now it is ready to review any intermediate designs and the evolution histories. Click on the  button to view the histories of the volume fraction and the mean compliance as shown in Figure 9.

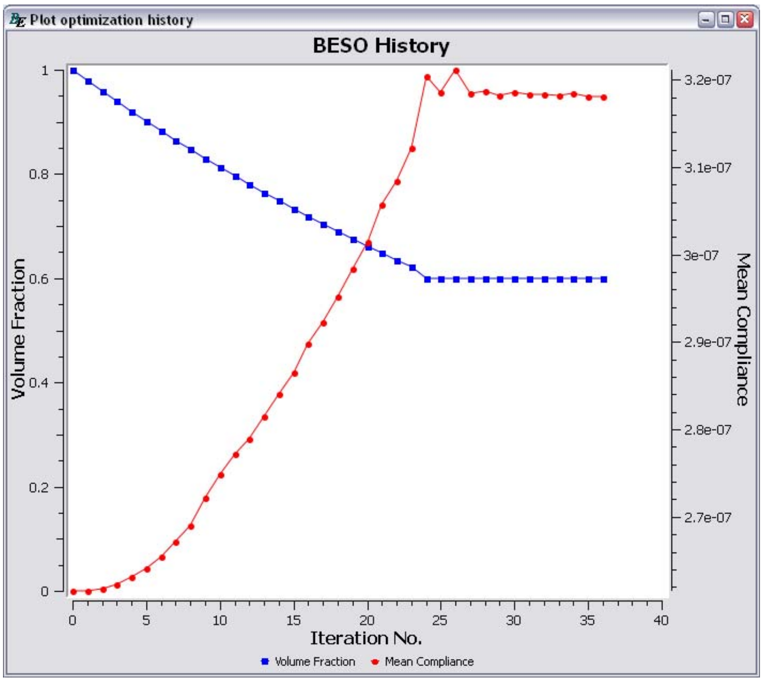


Figure 9 Evolution histories of the volume fraction and the mean compliance.

- By clicking on the  button, the "Intermediate design" dialogue will appear. Select a desired iteration number and click "Show", the corresponding intermediate design will be displayed. The intermediate designs can be saved to picture files in ".png" or ".bmp" format using the  button in the toolbar area.
- Click the  button and select a frame display speed. A speed of 1 iteration/second is recommended for clearly showing the whole optimization process. Then click "Run" to start the movie.

7. Continuing Optimization from an obtained design

Sometimes the optimization run stops before the solution converges due to reasons such as that the maximum iteration number has been set too small. BESO2D offers the user an option to continue the optimization process from the design of a previous optimization run.

In this example, we demonstrate how to further optimize the Michell type structure from the previous solution and obtain a final optimal design with 20.0% volume fraction.

- Click on the  button in the “Draw/Run Toolbar”. In the “Start BESO” dialogue, input 20.0% for the objective volume fraction.
- Uncheck the “New optimization run” checkbox and click “OK”.
- After a short while, the optimization run will stop and the final design with 20.0% volume fraction will be obtained as shown in Figure 10.

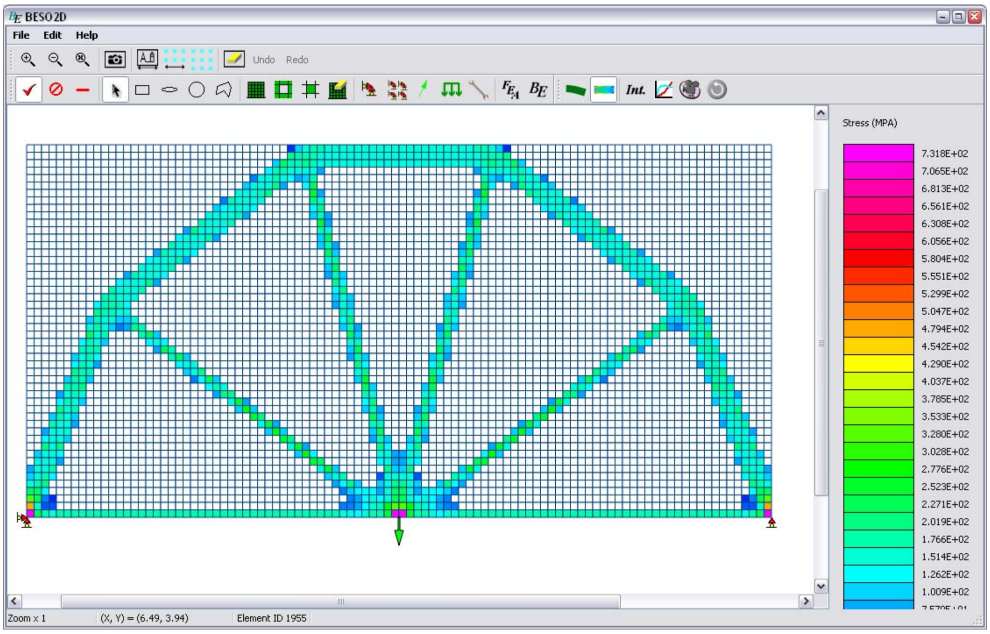


Figure 10 The final optimal design of the Michell type structure with 20.0% volume fraction.

- Follow the same steps in “6. Viewing the final optimal design and the evolution histories” to view the intermediate designs and evolution histories.

Getting started with Command Line usage of BESO2D

In this section, a step-by-step guidance for running the BESO2D from a command line is presented. This includes optimizing a structure from an initial design and continuing optimization from a previously obtained design solution. The file formats of the BESO parameter file and the model file accepted by BESO2D engine are introduced in *BESO Manual - Command Line Usage*.

1. Optimize a structure from an initial design

Before optimizing a structure using the BESO2D engine, one needs to have a working folder where we have full control, i.e. file reading, writing and deleting. Various files will be produced by BESO.exe including the model files of the intermediate designs, a "Result.txt" recording the evolution histories and a brief report in "Report.txt". In this section, it is assumed that the working folder is "D:\BESO2D\".

- Edit the model file of the initial structure and the parameter file (manually or using a program such as BESO2D GUI or a third-party software package). In this section, it is assumed that we have the model file named "Test.txt" for the previous initial Michell type structure in section "Getting started with GUI usage of BESO" in this manual. The parameter file "Parameters.txt" used in this example contains the following contents.

```
*Model<BMesh>
*EvoVolRatio<0.02>
*FilterRadius<0.3>
*ConvTolerance<0.001>
*MaxIter<200>
*ObjVolFraction<0.6>
```

  Now we can start optimizing the structure by calling BESO2D in a command line.

- Empty the working folder "D:\BESO2D\". Put the BESO2D engine "BESO.exe", "Test.txt" and "Parameters.txt" into the working folder.
- Open the command line window: click on "Run..." in the Windows Start menu, input "command" in the edit field and then click on "OK" button.
- Change the current directory to the folder where "BESO.exe", "Test.txt" and "Parameters.txt" have just been stored by typing "d:", then "cd BESO2D" in the command line window as shown in Figure 11.
- Under the prompt "D:\BESO2D>", type "BESO Test.txt Parameters.txt" and press "Enter" key. Now the program starts running.

Figure 11 Running BESO2D engine from a command line.

- Some information will be displayed in the command line window, e.g. the iterations completed. When the program stops due to the maximum iteration number being reached or the solution having converged, a message "END OF BESO" will be displayed in the command window.
- Now the final design can be imported to BESO2D GUI to be viewed. Under the "File" menu of the BESO2D window, click "Import Model", go to the working folder "D:\BESO2D" and select the model file with the largest iteration number which contains the last design. Click on "OK" to open this design. Then the final design will be displayed in the BESO2D GUI window as shown in Figure 12. Note that the designs in the last few iterations are similar due to the convergence of the solution.

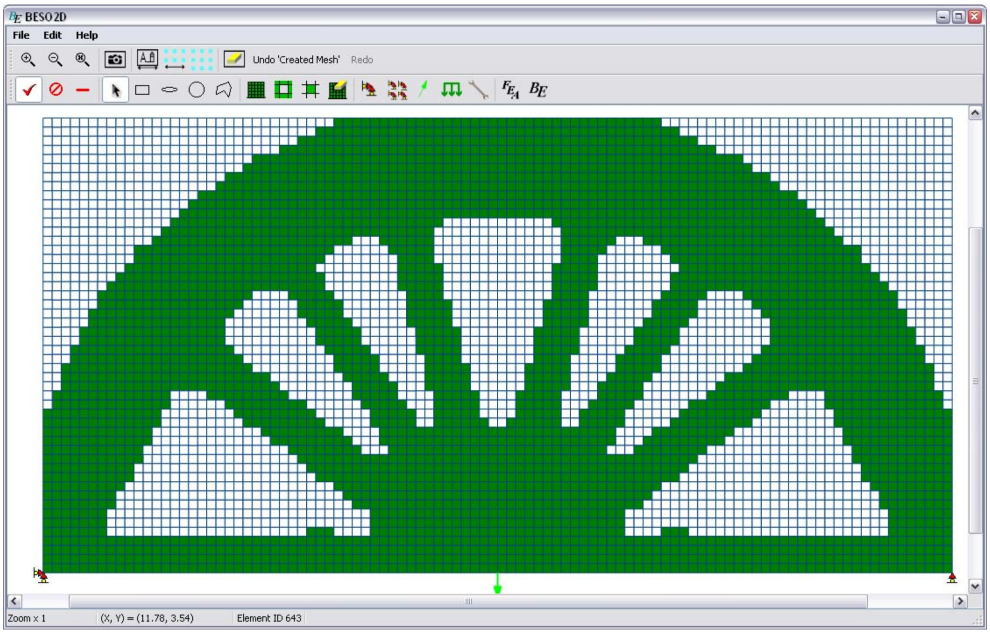


Figure 12 Displaying the solution with 60.0% volume fraction obtained from a command line optimization.

- View the evolution histories of the volume fraction and the mean compliance of the structure by opening the "Result.txt" in the working folder.

- A brief report showing the total number of iterations used and the procedure termination criterion can be found in the text file “Report.txt” in the working folder.

2. Continuing optimization from a previously obtained design solution

After an optimization run is finished and an optimal design is obtained, the structure can be further optimized using different optimization parameters. In this example, we change the objective volume fraction from 60.0% to 20.0% and further optimize the Michell type structure.

- Open the “Parameters.txt” in the previous working folder “D:\BESO2D\”, set a new objective volume fraction by changing the corresponding line to *ObjVolFraction <0.20>. Save and close this file.
- Under the command prompt “D:\BESO2D>”, type the same command as before: “BESO Test.txt Parameters.txt”. Press enter to start the optimization further. Now the BESO2D engine will automatically continue the optimization process from the end design of last run. Again a message will appear once the optimization is finished.
- Open the BESO2D GUI window, import the model file of the final design to view the final optimal design as shown in Figure 13.

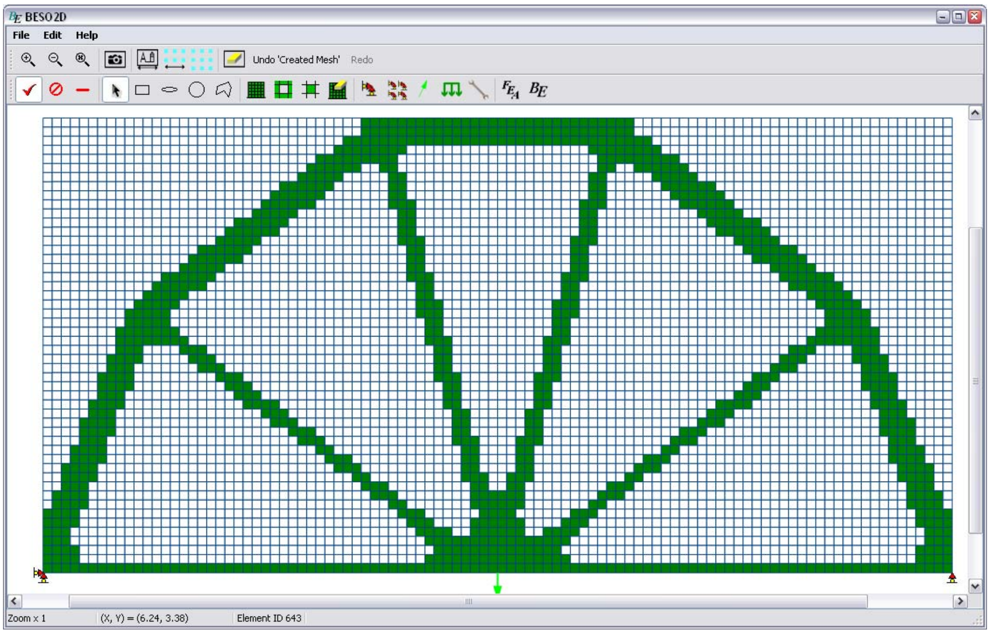


Figure 13 Displaying the solution from command line optimization for the new objective volume fraction of 20.0%.

- View the evolution histories by opening the “Result.txt” in the working folder “D:\BESO2D”. The additional result data from the continued optimization run are appended to the previous results.

- A new report is found in “Report.txt”.

If the user has any difficulties in obtaining or using BESO2D, or has queries about updates of the program, please contact us by email at the following address:

Mike.Xie@rmit.edu.au (Y.M. Xie)