



Engineering Technology Associates, Inc.

---

# **ETA / VPG Batch Meshing User Guide**

## **Version 1.0**

**December 2006**

---

Engineering Technology Associates, Inc.  
1133 East Maple Road  
Troy, Michigan 48083 USA

## **eta/VPG Batch Meshing User Guide**

---

### **Introduction**

The goal of creating a so-called “batch meshing software” is to allow engineers to create a finite element mesh of a surface or set of surfaces without opening a software toolset with complicated menus, thereby streamlining the mesh creation process.

The eta/VPG Batch Meshing software was created with this goal in mind. Users may import CAD data directly into the software, define the meshing parameters, and generate a finite element mesh without opening any visualization tools and without using a general-purpose finite element modeling user interface.

### **About the eta/VPG Batch Meshing Software**

The Batch Meshing software uses the Topology Automeshing algorithm which is found inside the eta/VPG software product. The Topology Automesher uses a concept of “parts” to create a finite element mesh on a surface or a series of surfaces.

The input data required for use of the Batch Mesher is CAD data, in IGES format. This format is directly imported into the Batch Mesher, and is then used to create the mesh. Users may input a single IGES file or a series of IGES files which will be used in the mesh creation.

Users have the option to view the CAD surface data which is to be meshed. The viewing of the CAD surfaces is performed by the creation of a #D data viewing file (E3D file) which is then viewable and can be manipulated within ETA’s 3D Viewer software, included in the Batch Mesher.

Mesh Quality parameters are specified by the user, and are used to guide the Topology Automesher to create a mesh which meets these parameters. Users should know that these are not guaranteed values, but guidelines by which the mesh is created.

Mesh size and CAD data healing parameters are defined by the user, and used by the Topology Automesher to define the nominal mesh size, and to evaluate the surface data for meshing operations.

Upon completion of the mesh, the software can optionally prepare a summary report of the mesh, noting the number of element, number of quadrilateral elements and number of triangular elements. This report can be saved in a text format for documentation of the model.

The resulting mesh can be viewed, similar to the CAD data viewing, using the 3D Viewer application.

Output files from the Batch Mesher are in the form of NASTRAN bulk data files, and include only node and element definitions. Part Ids are included as a way to differentiate the various CAD data parts within the model.

### Using the eta/VPG Batch Mesher

To execute the Batch Meshing software double click the ETA Batch Mesh icon on the desktop.



The software opens a single page user interface

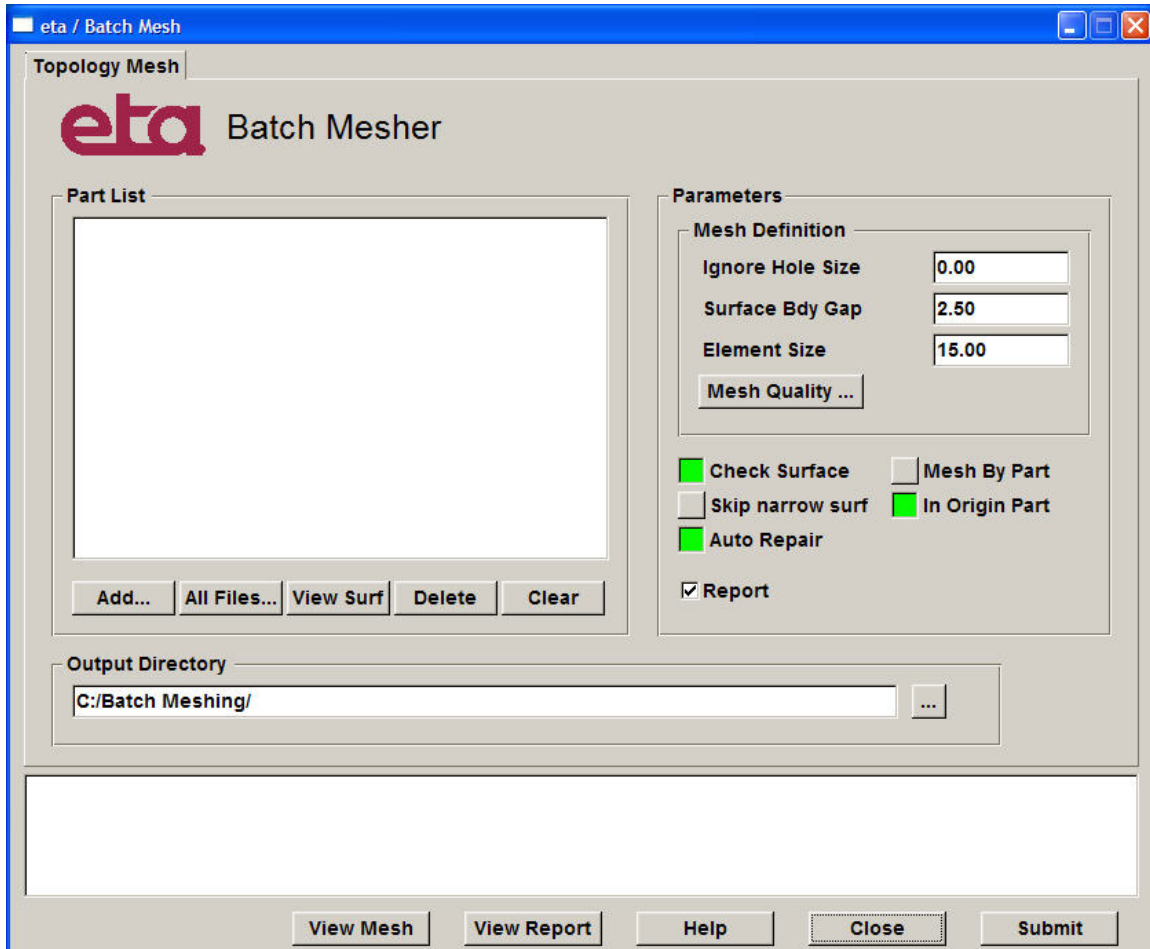


Figure 1: eta/VPG Batch Mesh User Interface

The user will note various areas on the user interface:

**Part List:** This contains a list of IGES files which will be meshed

**Add...** : When selected, this option opens a dialogue window allowing the user to select the IGES files to be included in the meshing operation. After the files are selected, they will appear in the Part List window.

## eta/VPG Batch Meshing User Guide

---

**All Files...** : This option allows the user to select all files within a directory, The file names will be displayed in the Part List window after selected.

**View Surf:** This command launches the eta 3D Viewer Application, and creates a temporary E3D dataset which is displayed in the 3D Viewer. All surfaces in the File List will be displayed.

**Delete:** The selected files in the Part List window will be deleted from the meshing operation when this function is selected.

**Clear:** Removes all CAD files from the Part List Window

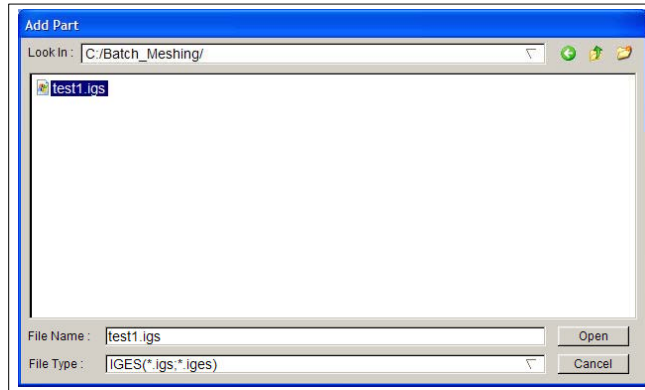


Figure 2 : File Selection Dialogue Window

**Parameters:** These quantities determine the mesh size and CAD data validation values to be used. Additional options allow for the software to perform additional data checks and provides for options to create the mesh file(s).

Under Mesh Definition the parameters are:

**Ignore Hole Size:** This determines the threshold value for inclusion of holes in the geometry. All holes with a diameter below this value will be ignored.

**Surface Bdy Gap:** This parameter specifies the maximum gap overlap allowed between adjacent surfaces. The Batch Mesher software will automatically repair any surface gap or overlap, which is below this specified value.

**Element Size:** This value is the nominal element size used in the meshing of all parts located in the Part List. Please note that elements both larger and smaller may appear in the mesh, regardless of the element size specified. Also, please note that the element size must be consistent with the geometry definitions for an appropriate mesh to be created.

**Mesh Quality:** When this button is selected, a panel will open, displaying all of the typical mesh quality parameters, and their default values by which the mesher will use in the mesh creation. The user may modify any of these values prior to executing the meshing operation.

**Check Surface:** Performed the Boundary Gap/Overlap Check

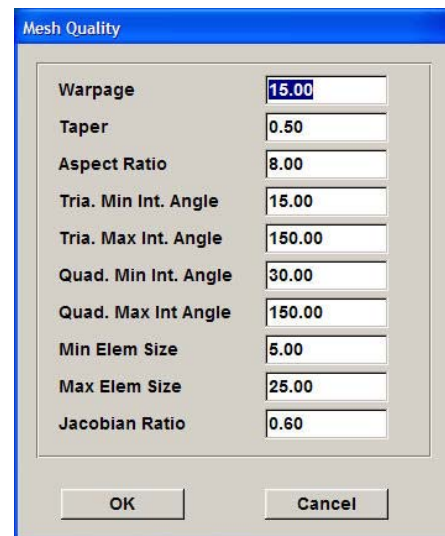


Figure 3: Mesh Quality Parameters

## eta/VPG Batch Meshing User Guide

**Skip Narrow Surface:** This option allows users to skip the meshing of long, slender (narrow) surfaces. The inclusion of these surfaces into the mesh may sometimes result in a poor quality mesh or a significant amount of very small elements.

**Mesh by Part:** when selected, ignores additional mesh options and continues the meshing operation.

**In Origin. Part:** This option specifies how part Ids will be created in the mesh. If this option is selected, each CAD file will have a corresponding part in the output file.

**Auto-Repair:** performs a mesh optimization operation prior to finalizing the mesh.

**Report:** When selected, this option creates a report which provides statistics about the mesh generated

**Output Directory:** This specifies the location of the resulting mesh data files. The user may select the target directory for the output files by selecting the “...” button. When selected a dialogue window will open to where the output directory may be specified.

**Message Area:** Messages pertaining to the meshing process, error messages and input prompts are displayed in the Message Area, directly below the Output Directory area.

**View Mesh :** This command launches the eta 3D Viewer Application, and creates a temporary E3D dataset which is displayed in the 3D Viewer. The user may view the mesh created in the meshing operation, and dynamically rotate/zoom, turn parts on/off and shade the mesh.

**View Report:** When selected, this option opens a window, displaying the contents of the report.

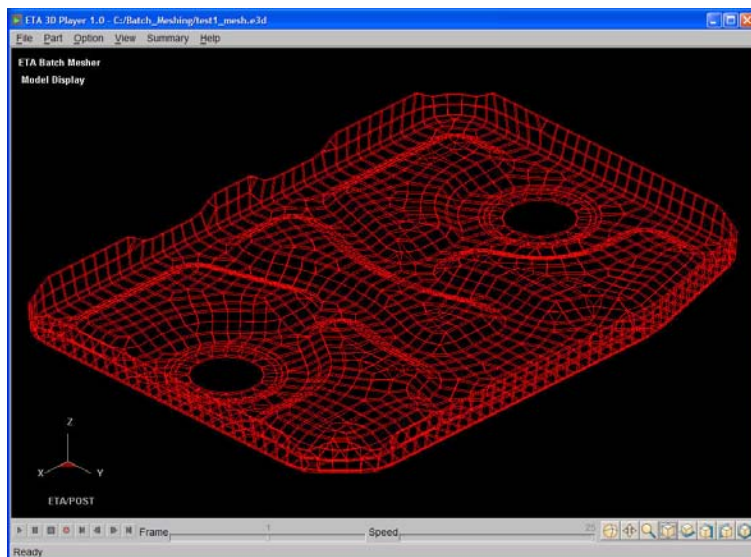


Figure 4: 3D Viewer Application Displaying Mesh



Figure 5: Report Results

## eta/VPG Batch Meshing User Guide

---

**Help :** This option is not currently operational.

**Close:** When selected, this function closes the ETA Batch Mesher software.

**Submit:** The files found in the File List will be submitted to the Topology Automeshing algorithm, using the parameters and mesh quality requirements prescribed. Messages relating to the progress of the meshing will be displayed in the Message Area during the operation. A message "Operation Complete" will be displayed when all files have been meshed.