

# Auto-Loading Fabrication Output into the CAM Editor

When generating Gerber, ODB++, NC Drill or IPC-356-D output, you can specify that one or more of these types of output be automatically imported into a new CAM Editor document (.cam). This is performed using the **Output Job Options dialog (Figure 1)**, accessed from the OutputJob Editor's \*Tools menu.

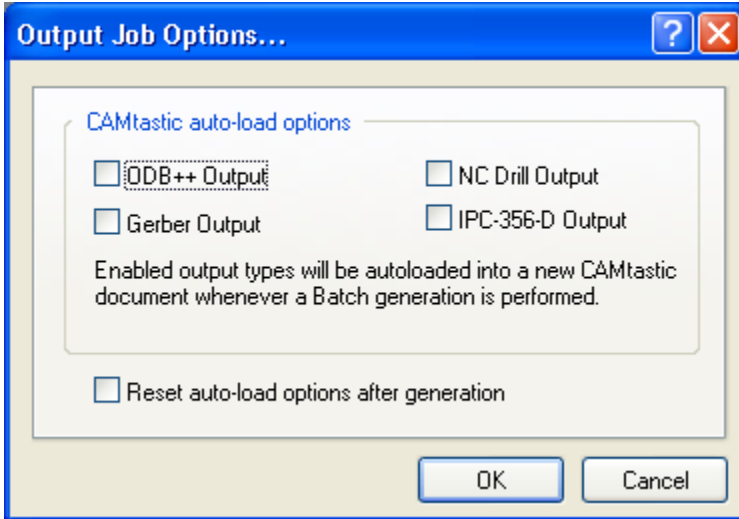


Figure 1. Configuring CAM auto-load options.

Enable the corresponding options for the outputs you wish to auto-load upon generation. Although you can enable one output type (e.g. Gerber) and run just the associated output generator, typically you would enable NC Drill, IPC-356-D and either Gerber or ODB++ and then run the associated Output Generators as a batch process. Once these options are defined they will persist. This means that the next time you run the Output Generators, the resulting output would be loaded into another new CAM Editor document. If you wish to be able to update just the existing CAM document, enable the **Reset auto-load options after generation** option. This results in the clearance (disabling) of all auto-load options after the initial generation. You can then gain access to the CAM Editor's **Rescan** and **Reload** commands (from the **CAM** panel), which perform time-stamp comparison of generated and existing (imported) files and loading of data into existing layers respectively.

## Related Topics

### [CAM Editor](#)