CONVERGENT

MANUFACTURING TECHNOLOGIES

COMPRO Plug-in
For
Abaqus
User's Guide

v1.0

October 2014

Convergent Manufacturing Technologies Inc.
6190 Agronomy Road, Suite 403, Vancouver, British Columbia, Canada, V6T 1Z3
Tel: 604-822-9682, Fax: 604-822-9659, inquiries@convergent.ca, www.convergent.ca

Table of Contents

Note: This guide is intended for qualified users who will exercise sound engineering judgment. The COMPRO Common Component Architecture (CCA) is inherently complex, and the examples and procedures outlined in this guide are not intended to be exhaustive or applicable to any particular situation. Users are cautioned to satisfy themselves as to the accuracy and suitability of the results of their analyses.

Convergent Manufacturing Technologies, Inc. (Convergent) will not be held responsible for the accuracy or usefulness of any analysis performed using the COMPRO software or the procedures, examples, or explanations in this manual. Convergent shall not be responsible for the consequences of any errors or omissions that may appear in this manual.

CONVERGENT MAKES NO REPRESENTATIONS OR WARRANTIES, EITHER EXPRESS OR IMPLIED, WITH RESPECT TO THE GUIDE AND SPECIFICALLY DISCLAIMS ANY IMPLIED WARRANTY OF NON-INFRINGEMENT OR MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. CONVERGENT SHALL IN NO EVENT BE LIABLE FOR ANY LOSS OF PROFITS, BE THEY DIRECT, CONSEQUENTIAL, INCIDENTAL, OR SPECIAL OR OTHER SIMILAR OR LIKE DAMAGES ARISING FROM ANY DEFECT, ERROR OR FAILURE TO PERFORM WITH RESPECT TO THE GUIDE, EVEN IF CONVERGENT HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THE RECIPIENT HEREBY ACKNOWLEDGES THAT IT HAS BEEN ADVISED BY CONVERGENT TO UNDERTAKE ITS OWN DUE DILIGENCE WITH RESPECT TO THE GUIDE.

IN NO EVENT SHALL CONVERGENT BE LIABLE FOR SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES, INCLUDING THOSE DAMAGES ARISING FROM LOST PROFITS IN THE USE OR MISUSE OF THE GUIDE, EVEN IF CONVERGENT HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

This guide is intended to assist with the process design and is not a substitute for independent testing of a product's stress, safety and utility. Convergent does not warrant the accuracy of the results obtained through use of the guide.

The COMPRO Common Component Architecture (CCA) software described in this guide is available only under license from Convergent Manufacturing Technologies, Inc., and may be used or reproduced only in accordance with the terms of such license.

This guide and the software described in the guide are subject to change without notice.

TRADEMARKS:

The trademarks and service marks (trademarks) in this guide are the property of Convergent Manufacturing Technologies, Inc. or third parties. You are not permitted to use these trademarks without the prior written consent of Convergent Manufacturing Technologies, Inc.

The following are trademarks of Convergent Manufacturing Technologies, Inc.: COMPRO, COMPRO Editor, COMPRO Common Component Architecture, COMPRO 2D, COMPRO 1D, COMPRO 0D.

Other company, product, and service names may be trademarks or service marks of their respective owners.

Convergent Manufacturing Technologies, Inc.
403-6190 Agronomy Road
Vancouver BC V6T 1Z3
CANADA
Tel: +1-604-822-9682
Fax: +1-604-822-9659
Email: support@convergent.ca

WWW: http://www.convergent.ca

1 Introduction

This guide outlines the features of the COMPRO Plug-in for Abaqus, and describes the suggested workflow for setting up and running a composites processing model in Dassault Systemes Simulia ABAQUS and Convergent Manufacturing Technologies COMPRO. There are three distinct analyses in composites process modeling: Thermo-Chemical analysis, Flow-Compaction analysis and Stress-Deformation analysis. This plugin can be used to accelerate the setup and run for all three analysis types.

Throughout this guide, some basic familiarity with ABAQUS/Standard and ABAQUS CAE is assumed. The user is referred to the ABAQUS User's Manuals for complete detail on using ABAQUS CAE and / or ABAQUS keywords. It should be noted that this guide was written with reference to ABAQUS v6.12. Other version of ABAQUS and ABAQUS CAE may differ from the guides given here.

It should be noted that a process simulation using COMPRO can be performed without the COMPRO plug-in. For such cases, please refer to the COMPRO modeling guidelines for instructions to manually set up a COMPRO analysis

2 Workflow:

The workflow for setting up a process simulation includes using built-in ABAQUS CAE features along with automations performed by the COMPRO plug-in. The COMPRO plug-in automates many of the commonly performed steps and guides the user in setting up the model. The COMPRO plug-in also makes many changes to the input files when a job is submitted. These changes allow for, for example, a single model to be set up while up to three analyses are actually run.

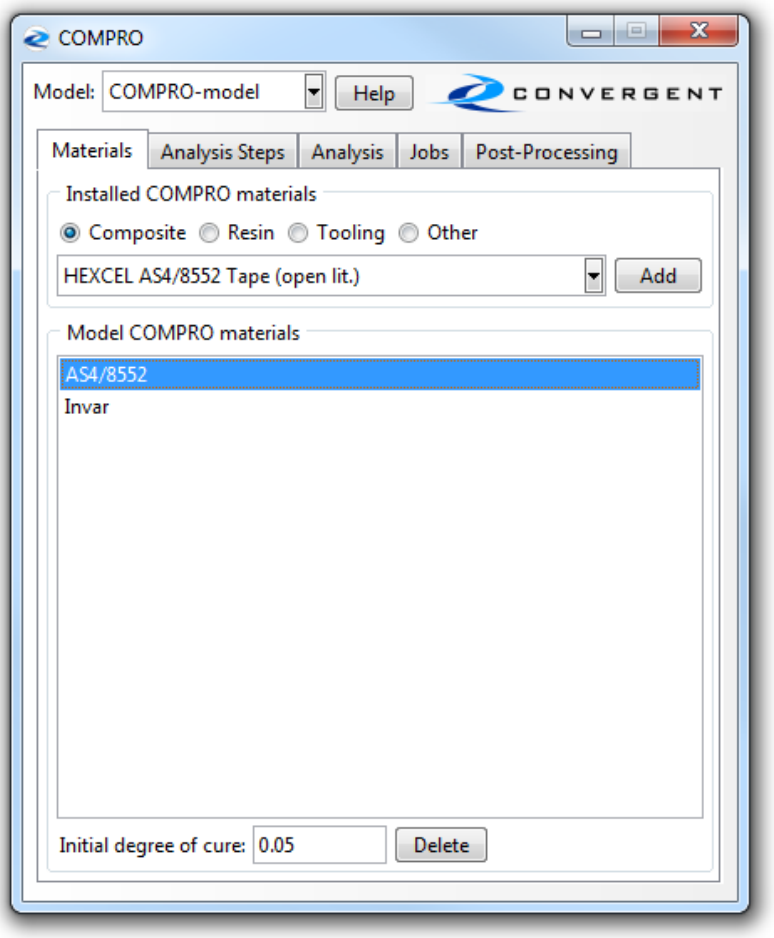
A typical workflow using the COMPRO plug-in would include:

Workflow Step	Primary User Interface
1. Create the part and tool geometry	ABAQUS CAE
2. Add materials	**COMPRO plug-in**
3. Create and assign sections	ABAQUS CAE
4. Assign material orientations	ABAQUS CAE
5. Create the assembly	ABAQUS CAE
6. Define the analysis steps	**COMPRO plug-in**
7. Assign initial and boundary conditions	ABAQUS CAE
8. Mesh the geometry	ABAQUS CAE
9. Run the job	**COMPRO plug-in**
10. Post-process the results	ABAQUS CAE / **COMPRO plug-in**

3 Plugin Features

3.1 Materials

The materials tab is used to add COMPRO materials to an ABAQUS model, adjust the initial degree of cure of COMPRO materials, and delete COMPRO materials.



3.1.1 Installed COMPRO materials

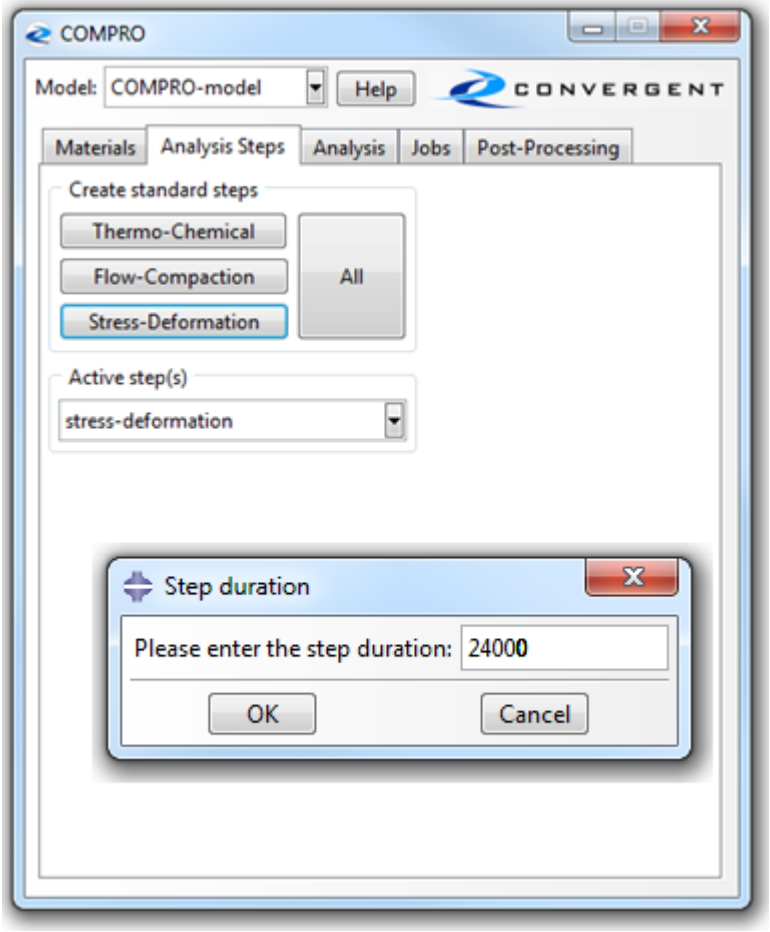
Installed COMPRO materials are organized based on material type (composite, resin, tooling and other). To add a material, select the desired material type, then select the desired material from the drop-down, and click the **Add** button.

3.1.2 Model COMPRO materials

This list includes all COMPRO materials in the selected ABAQUS model. After selecting a material from the list, you can modify the initial degree of cure of that material or delete it. COMPRO materials can also be removed from the model from CAE.

3.2 Analysis Steps

The Analysis Steps tab is used to create the standard steps used in process simulation for composites, as well as easily activate one of these steps in the current model.



3.2.1 Create standard steps

Clicking “Thermo-Chemical”, “Flow-Compaction” or “Stress-Deformation” buttons will add that respective analysis step to the model. For the stress deformation button, two steps are added to the model, one for the cure cycle and one for tool removal. Individual analysis steps may be added or the “ALL” button will add all three steps at once.

3.2.1.1 Step duration

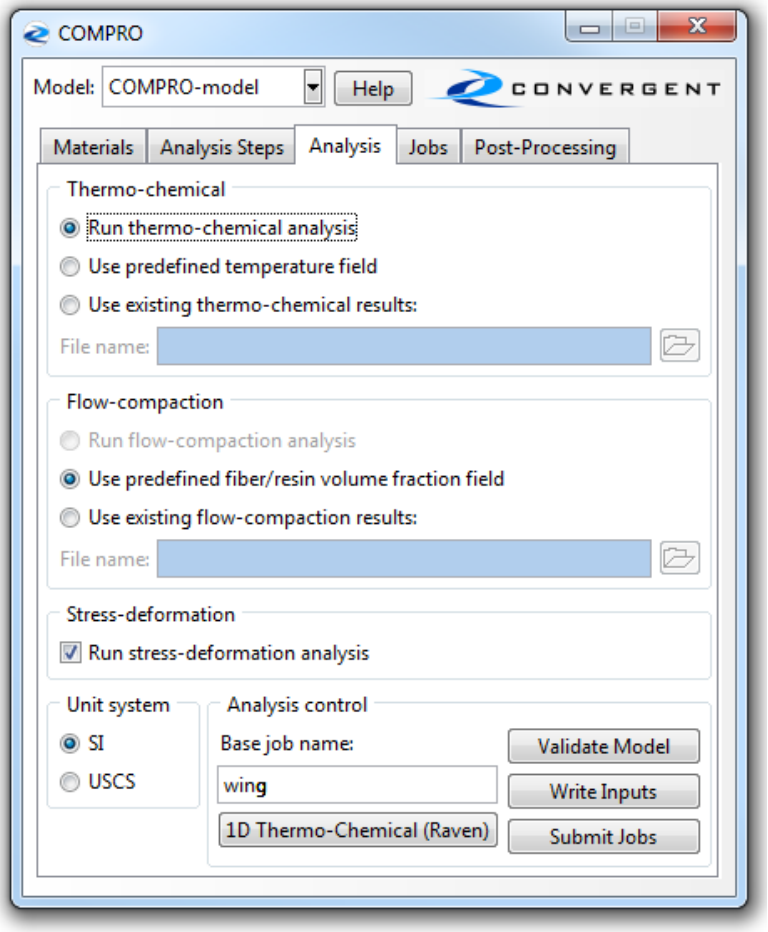
Clicking any of the analysis type buttons will prompt the user for the step duration. This is the cure cycle step time, usually given in seconds (depending on the units used in the model).

3.2.2 Activate step(s)

In order to setup the boundary conditions and other aspect of each step, it is necessary to suppress the other steps and resume the one currently being modified. To streamline this process, you can select the desired analysis type from the **Active step(s)** drop-down and the appropriate step(s) will be activated, with others supressed.

3.3 Analysis

The Analysis tab is used to select the analyses to include in the job, modify the base job name, select the model unit system, validate the model, write input files, and submit jobs.



3.3.1 Thermo-chemical

3.3.1.1 Run thermo-chemical analysis

Select this option to include a thermo-chemical analysis with the job. This option requires a thermo-chemical step with appropriate boundary conditions applied to be defined in the current model. If subsequent flow-compaction and/or stress-deformation analyses are selected, the results from this thermo-chemical analysis will be fed into those analyses.

3.3.1.2 Use predefined temperature field

This option requires a predefined temperature field to be defined in CAE for the flow-compaction or stress-deformation analyses, depending on which are being run.

3.3.1.3 Use existing thermo-chemical results

This option will read temperature results from an existing thermo-chemical ODB which must be compatible with the current model. The results from the selected thermo-chemical analysis will be fed into subsequent flow-compaction and/or stress-deformation analyses.

3.3.2 Flow-compaction

3.3.2.1 Run flow-compaction analysis

Select this option to include a flow-compaction analysis with the job. This option requires a flow-compaction step with appropriate boundary conditions applied to be defined in the current model.

3.3.2.2 Use predefined fiber/resin volume fraction field

This option will use the uniform fiber / resin volume fraction as defined in the material file for the entire part for the stress-deformation analyses. If desired, the user can manually define a fiber/resin volume fraction field by modifying the seventh value of the"*initial conditions" keyword in the input file for specific elements or element sets.

3.3.2.3 Use existing flow-compaction results

This option will read resin volume fraction results from an existing flow-compaction ODB which must be compatible with the current model. The results from the selected flow-compaction analysis may be fed into subsequent stress-deformation analyses.

3.3.3 Stress-deformation

3.3.3.1 Run stress-deformation analysis

Select this option to include a stress-deformation analysis with the job. This option requires a stress-deformation step with appropriate boundary conditions applied to be defined in the current model.

3.3.4 Unit system

COMPRO materials are typically defined using SI units. If your model is defined using USCS units, you can change the selected unit system to have the COMPRO material properties converted appropriately.

3.3.5 Analysis control

3.3.5.1 Base job name

The base job name is combined with the various analysis types to name jobs and generated INP files.

3.3.5.2 1D Thermo-chemical (Raven)

Clicking the **1D Thermo-chemical (Raven)** button will prompt you to select an element face. After selecting a face, a stack representing the geometry, materials, layup and boundary conditions for a 1D drill through below the selected element face will be passed to Raven.

3.3.5.3 Validate

The **Validate** button will perform a series of checks on the model to warn you of any missing information or possible issues with how the model is defined.

3.3.5.4 Write Inputs

The **Write Inputs** button will write all of the required files to run the selected analyses to the current work directory, but will not submit the job. These files include:

- Input files (**.inp*) for all selected analysis types
- COMPRO log files (**.clog*) all selected analysis types
- The *cca-configuration.xml* file
- The *abaqus_v6.env* file if it does not already exist
- If temperatures are transferred from a thermo-chemical analysis to a flow-compaction analysis, a *generateTemperatureINP.pyc* file
- If fiber / resin volume fraction is transferred from a flow-compaction analysis to a stress-deformation analysis, a *generateVfINP.pyc* file
- A *batch-job-name.bat* file and a *batch-job-name.sh* file

The *generateTemperatureINP.pyc* code is run after a thermo-chemical analysis but before a flow-compaction analysis. It generates a file named *temperature-jobname.inp* which contains keywords that ensure that the temperature history is properly transferred from the thermo-chemical analysis to the flow-compaction analysis.

The *generateVfINP.pyc* code is run after a flow-compaction analysis but before a stress-deformation analysis. It generates a file named *vf-jobname.inp* which contains keywords that define the initial fiber / resin volume fractions for each element in the stress-deformation analysis.

The *batch-job-name.bat* and *batch-jobname.sh* files are batch run files formatted for Windows and Linux operating systems respectively. These files list the sequential commands required to submit the job based on the selections given in the COMPRO plug-in.

To submit the job files, open a command window in the work directory and run the commands: *batch-job-name.bat* or *bash batch-jobname.sh* depending on if your operating system is Windows or Linux respectively.

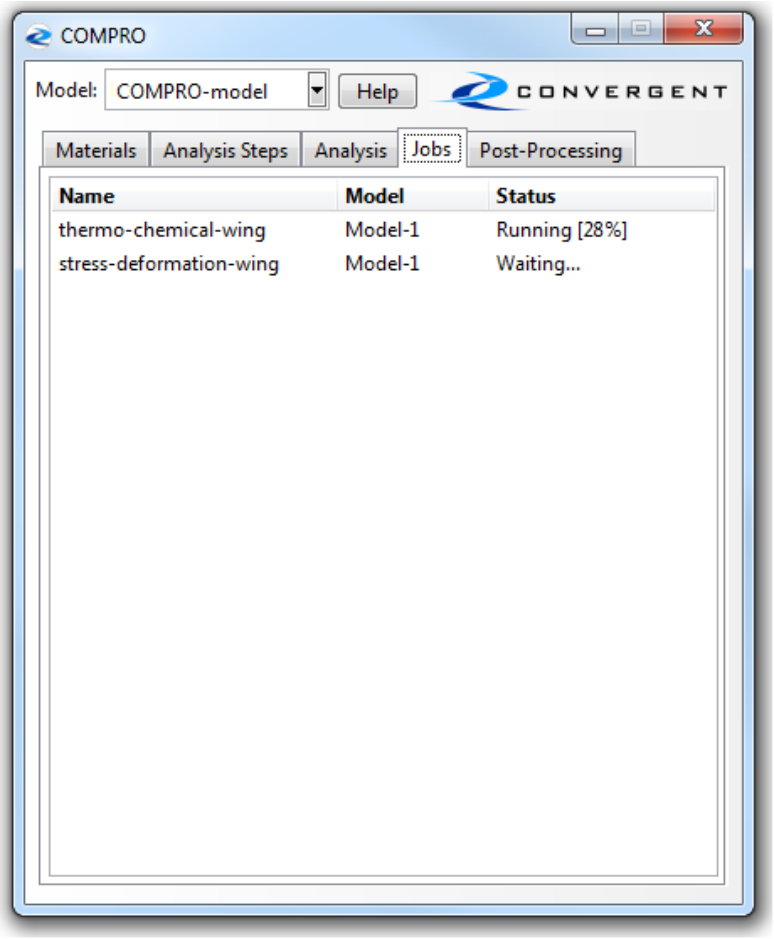
3.3.5.5 Submit Jobs

The **Submit Jobs** button will write INP files and cca-configuration.xml files and run each analysis in sequence. If selected, the thermo-chemical analysis will be run first, followed by the flow-compaction second and stress-deformation analyses third.

3.4 Jobs

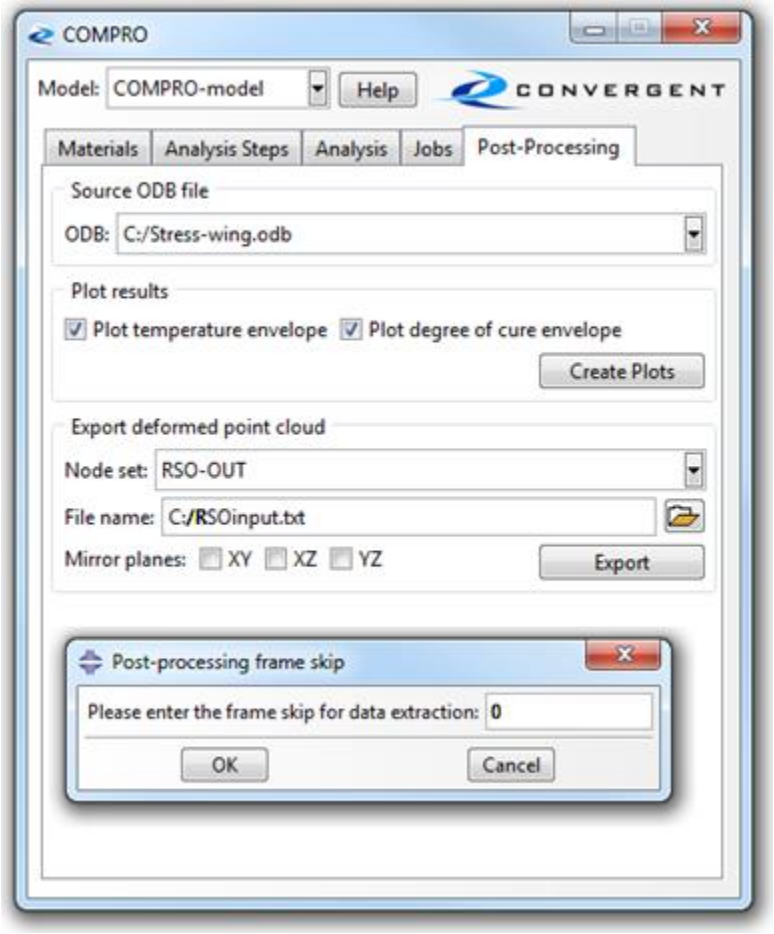
The Jobs tab lists jobs created and submitted by the **Submit Jobs** button on the Analysis tab. This provides a quick way to check on the progress of submitted jobs. The ABAQUS CAE job manager can be used to retrieve more information about the status of the jobs.

Note that in order for flow-compaction and stress-deformation analyses to be automatically submitted after the associated thermo-chemical analysis is complete, you must leave the COMPRO plugin visible, and must not close the associated CAE file.



3.5 Post-Processing

The Post-Processing tab provides tools to automate certain commonly performed post-processing tasks.



3.5.1 Source ODB file

In order to use post-processing tools, an appropriate ODB must be open in ABAQUS CAE. The desired ODB can then be selected in the **ODB** dropdown.

3.5.2 Plot results

Temperature and degree of cure envelopes are useful plots for process simulation of composites. Both plots can be generated using built in tools within the ABAQUS visualization module, however it can be time consuming and non-intuitive to do so. The plot results tools make it easier to extract these results.

With an ODB open and selected, clicking **Create Plots** will extract the selected envelope types and create appropriate plots. The tool will query the temperature and/or degree of cure for the entire model at every time increment. Plots are then created with the maximum and minimum temperature and/or degree of cure curves.

3.5.2.1 Post-processing frame skip

Extracting the necessary data from the ODB to create plots can be a time consuming operation especially for large models or with long step times. Clicking **Create Plots** will prompt the user to enter the frame skip for data extraction. Increasing the value entered here will decrease the amount of data extracted from the ODB and speed up plot generation. A value of 0 will skip no frames and extract all the data.

3.5.3 Export deformed point cloud

A deformed point cloud can be useful when adjusting tool geometry based on analysis results. With a stress-deformation ODB open and selected, select a node set of interest (usually all the nodes on the bottom of the part), specify an output file name, and click **Export** to extract the data. Note that depending on the size of the model this operation may take some time.

The TXT file that is created is formatted to be compatible with the "digitized morphing" feature in Dassault Systems CATIA. This feature is part of the realistic shape optimizer (RSO). For each node in the node set, a line is written to the text file that contains six numbers. The first three numbers are the base coordinate of the node in the global XYZ space. The last three numbers are the relative deformation the node also in the global XYZ space.

The mirror planes checkboxes can be used to make copies of the deformed points mirrored about the primary coordinate axes. Each checkbox checked will create another set of copies, thereby doubling the number of points in the exported file. This can be useful if the Abaqus model represents a symmetric subset of the full model.