



ESAComp 4.6 RELEASE NOTES

Version 4.6.0.040, March 15, 2017

PYTHON SCRIPTING INTERFACE

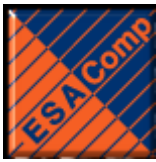
- **Python scripting interface enables easy access to customization and extension of existing analyses.** Python scripting interface replaces the existing CLIPS interface. Micromechanics and various failure criteria have already been re-implemented in Python (see Python folder of ESAComp installation). Other examples include load response/failure calculation of structural elements and the newly added laminate first ply failure strength in cylindrical bending.
- **Batch files** can be read from graphical user interface (via File -> Run batch file) or command line:
`C:\Program Files (x86)\ESAComp 4.6.0\ESAComp46.exe" -b script.py`
- In the 4.6 version the CLIPS interface works still besides the Python interface, but will be retired in the future versions.

PANEL/CYLINDER ENHANCEMENTS

- **Shell element based modeling of stiffeners in panel analysis** has been implemented and it is the default setting. Beam element based stiffeners can still be used as well (selected from Analysis Options).
- **Edge load types bending and twisting** have been added for panels.
- **Automatic mesh adaptation for cylindrical shells** is available through Analysis Options. The mesh parameters are selected so that the accuracy of the analysis and the solution time are in balance. For more information see Example Case EC29.

COMPOSICAD-ESACOMP INTERFACE FOR COMPOSITE PRESSURE VESSELS (CPV)

- **Solid element based analysis of CPVs**
 - The new module utilizes input from CompositaD filament winding process simulation tool (www.compositcad.com) by the Seifert and Skinner Group. ESAComp adds the structural simulation capability in the design process.
 - The element mesh, definitions for material ID and orientation at each element and the pressure load are imported from CompositaD to ESAComp and combined with material data from ESAComp. A solid based FE model of the structure is created, solved and post-processed in consecutive automated steps.
 - The cylindrical shell add-on module license is required for using the CompositaD interface.
 - See Example Case EC43 for further information.



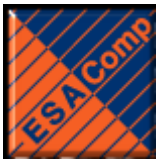
- **Enhancements to shell based analysis of CPVs**
 - Instead of the whole model specific range of laminates can be selected for post processing in the 3D results viewer. With this capability one can make efficiently local studies.
 - While creating a laminate load vector for the selected element associated laminate is also created. With this approach, different laminate sections of the CPV model can be used in other ESAComp analyses.
 - The area corresponding to the open hole is considered by defining a compensating line load at the edge of the hole.
 - Line charts are available for the surface strains and stress resultants through "Print preview...". The results are provided for the selected range of elements.
 - Increased robustness against models with incorrect data (e.g. missing section data).

FE POST-PROCESSING FOR ALTAIR HYPERWORKS

- **ESAComp supports the new workflow using HyperWorks 14.130 or higher**, where ESAComp export and import are directly available from the Aerospace toolbar (use User profile "Engineering solutions" -> "Aerospace"), without having to install the ESAComp toolbar. The results can be imported and contoured using either HyperWorks' matrix browser functionality or reading the HWASCII file optionally created during import. The latter is well suited for viewing ply-wise results.
- **Automated post-processing for HyperWorks** import files has been enabled. Currently, two versions are supported for batch runs:
 1. -hwb used to execute the post processing
`"C:\Program Files (x86)\ESAComp 4.6.0\ESAComp46.exe" -hwb HW-export.csv`
 2. -hw used to import the file and open ESAComp in GUI mode
`"C:\Program Files (x86)\ESAComp 4.6.0\ESAComp46.exe" -hwb HW-export.csv`
- **Significant performance improvement** for FE post-processing of HyperWorks imports.
- **Use of plies from existing case** instead of ply data from the CSV file created HyperWorks. Hence, the full set of material data can be used for post-processing, e.g. transverse shear strength (missing in MAT8) and interlaminar shear strength.
- **Global ply IDs** have been introduced to ESAComp as an additional internal variable of each layer.

DATA BANK UPDATE

- **Core material update** including BALSA cores and FOAM cores; PEI, PES, PET, PMI, PUR, PVC and SAN.



DOCUMENTATION AND DEMO CASES

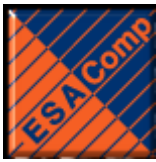
- **Six example cases** have been added covering shell and solid element-based simulation of CPVs, use of shell-based stiffeners in different types of panel applications, and utilization of the new panel load type - edge bending.
- **Python scripting interface functions and underlying data structure** have been documented in new HTML help (see Document/html folder). Three currently existing examples cover laminate creation, micromechanics, and failure criteria.
- **Updated guide “ESAComp-HyperWorks post-processing interface workflow.pdf”** has been added to the documentation. It illustrates the workflow, compares values for a reference model and explains the different notation of both programs.

OTHER ENHANCEMENTS

- **Laminate strength in cylindrical bending** is now available under laminate strength analysis (select the associate result macro from the drop-down menu). The analysis has been implemented with Python scripting and demonstrates the type of additions that could now also be accomplished by the end-users.
- **Notched strength carpet plots** have been implemented as an extension of the ply carpet plots by combining with the notched laminate failure analysis. The related input parameters, the notch radius (NC_radius) and edge distance (NC_edge_distance, the characteristic distance where the failure analysis is performed) are defined under extension variables (Edit -> Extension variables). The carpet plot specification dialog will be extended in the future versions for the definition of these input parameters.
- **The ANSYS APDL post-processing interface** that was formerly licensed as an add-on module is now available to all ESAComp users. See ...\\ESAComp 4.6.0\\Ansys\\README.txt for more information on the interface.
- The RLM based license management has been updated to RLM 12.0.
- Altair licensing updated to v13.1.0. (Concerns ESAComp use through Altair Partner Alliance only.)
- Numerous smaller enhancements.

FIXED BUGS

- In panel models boundary beam elements caused numerical instability in the 4.5 version in case of very small element dimensions. Boundary conditions are handled with boundary beam elements with finite stiffness. Stiffness is proportional to the size of the elements.
- For hat stiffened panels stiffness matrices of the leg laminates have been fixed.
- For hat stiffened panels the selection of the FEA reference plane now reflects to the height of the stiffener.



- Previous versions of ESAComp (prior to 4.6.0) and HyperWorks (prior to 14.130) could under certain circumstances produce wrong post-processing results, due a wrongly transferred/transformed orientation information.
- CriticalLayerOrientation in HyperWorks post-processing for negative orientations was missing in the corresponding column.
- In addition, many noncritical bugs have been fixed.