

Appendix V

The ESATAN-TMS Finite Element Analysis Method User Experiences

Gunnar Sieber Stefan Kasper
(Jena-Optronik GmbH, Germany)

Abstract

Based on a current space application analysis case, first-hand experiences of using the ESATAN-TMS Finite Element (FE) analysis method are presented. The steps from geometric model creation to post-processing of results are shown. Differences with respect to the traditional Lumped Parameter (LP) analysis method are highlighted and specific aspects related to the new FE analysis approach are discussed. Also suggestions for further improvement of this modeling approach are made.



The ESATAN-TMS Finite Element Analysis Method: User Experiences

G. Sieber & S. Kasper, 09.11.2011



space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 1



Introduction

The finite element analysis method offers new possibilities for thermal modeling and analysis with ESATAN-TMS

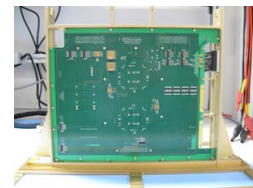
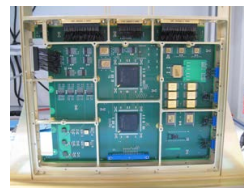
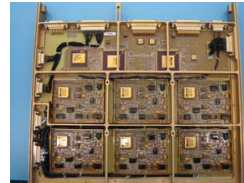
Motivation to use FE method in thermal analysis:

- need for data exchange with structural analysis, optical analysis, and mechanical designers → in most analysis fields, FE models are commonly used
- automated model generation (geometry) via CAD import is desirable

space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 2

Simulation Case: Electronics Box for Sentinel-2 (VCU)



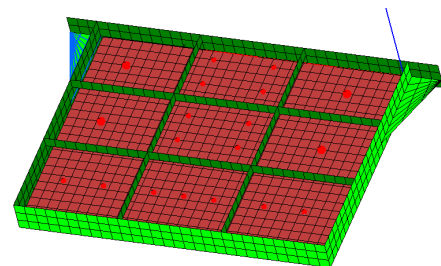
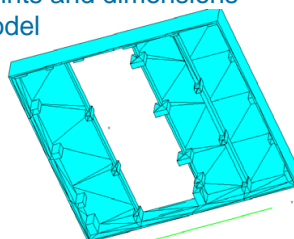
space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 3

Model Creation (1/2)

CAD Import using ESATAN-TMS CADConverter 1.0

- “simple shapes” converted to triangles → too many individual shells
- CAD 3D solids vs. ESATAN-TMS 2D shells
- converted geometry is used only as basis for model creation
- individual simple shape shells are created using coordinate points and dimensions of the converted model

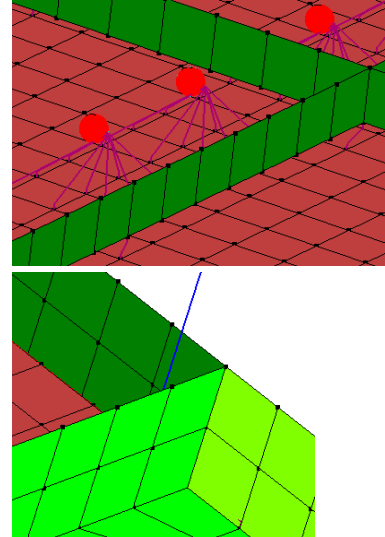


space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 4

Model Creation (2/2)

- use of fixed mesh size (e.g. 10 mm) allows automatic conductance generation at shell interfaces (fused interfaces)
- fully automated node numbering
- fused connections of shells can be avoided by introducing spatial gaps between shells
- definition of contact conductance only possible between shell edges: for T-shaped connections, separate shells at either side of the T-shaped connection must be defined

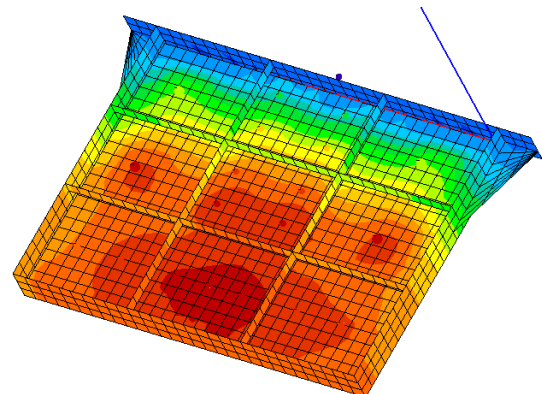
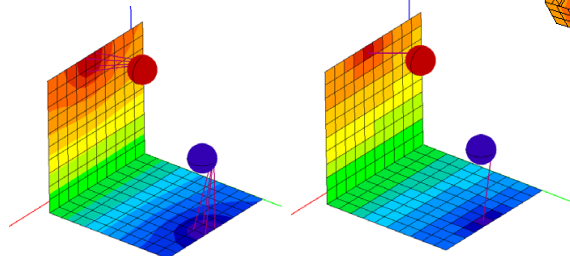


space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 5

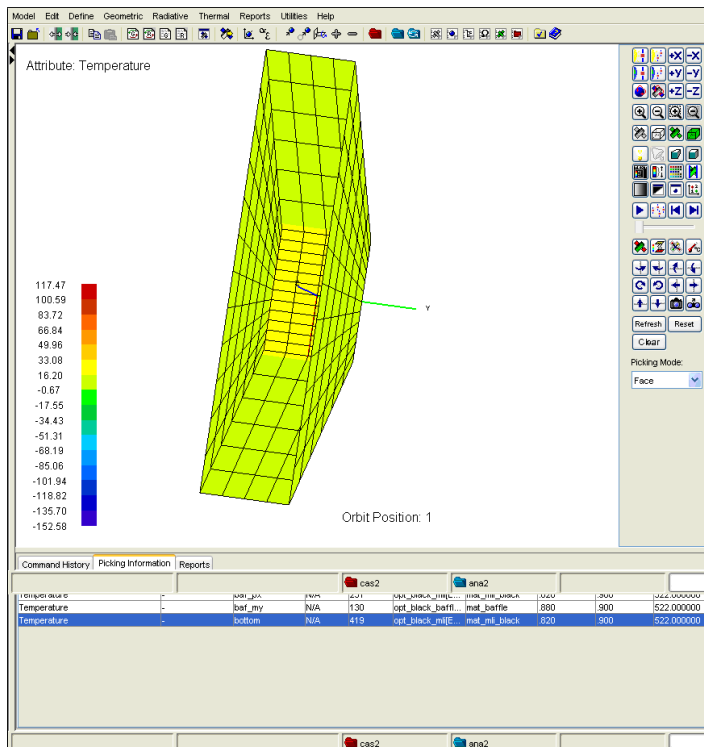
Post-Processing (1/2)

- ESATAN-TMS Workbench: for FE models, a more realistic visualization can be obtained via colored temperature maps compared to the classical LP method
- ThermNV: handling/performance problems due to very high number of nodes



space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 6

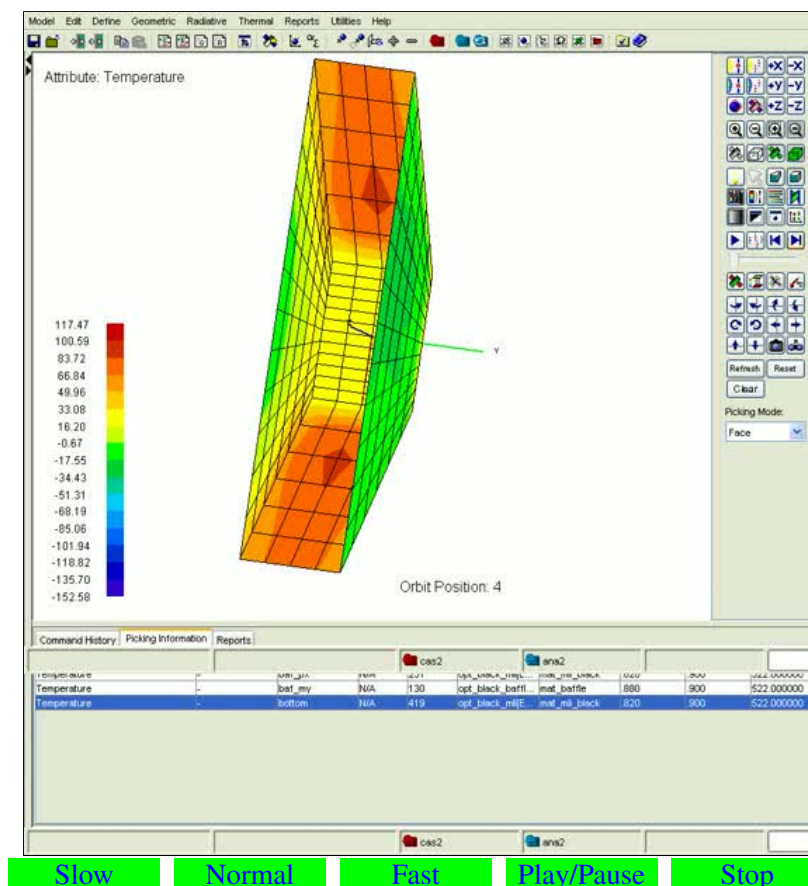


Post-Processing (2/2)

Transient in-orbit temperature variation (optical sunshield)

space for success

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 7

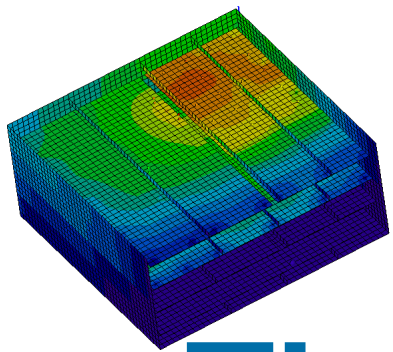


Open Issues related to FE modeling

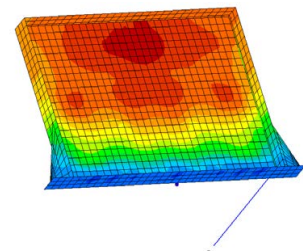
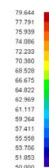
- CAD Import (3D solids → 2D shells), smart recognition of simple shapes, minimize number of individual shells
- Conductive interfaces: proper treatment of all interfaces (T-shaped connections, adjacent shells with different meshing, and cylindrical/circular interfaces)
- Post-processing of large models in ThermNV

 **space for success**

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 8



Thank you

 **space for success**

The ESATAN-TMS Finite Element Analysis Method: User Experiences, page 9

