Appendix F

SINAS: Generic tool for mapping lumped parameter temperatures to a finite element model for thermo-elastic analyses

Simon Appel
(AOES, The Netherlands)

SINAS

A generic tool for mapping lumped parameter temperatures to a finite element model for thermo-elastic analyses

Simon Appel
Analysis and Verification Section
ESA/ESTEC D/TEC-MCV
AOES Netherlands B.V., Leiden

esa

Mechanical Engineering Department
Thermal and Structures Division

Topics

- What is SINAS?
- History
- Overview of functionality (What it can do and what it not can do)
- The method
- Example
- How to get it

esa

Mechanical Engineering Department
Thermal and Structures Division

What is SINAS

- Suite of modules that convert/interpolate

Lumped parameter thermal node temperatures

↓

Finite element nodal and element temperatures

For thermo-elastic analysis

esa

Mechanical Engineering Department
Thermal and Structures Division

History

- Developed by Dutchspace (former Fokkerspace)
- Under a series of small ESA contracts from around 1985 until 1998
- Initiative from the ESA/ESTEC structural section
- IPR handed over to ESA in 2004
- Currently maintained by ESTEC TEC-MCV
  - No more planned development

esa

Mechanical Engineering Department
Thermal and Structures Division

Available information for mapping

- Thermal lumped parameter model side (in most cases):
  - Only radiative geometry (GMM)
    - Often does not coincide with structural geometry
  - No geometry for the conductive nodes
    - Have to rely on sketches in model description
- Structural finite element model side:
  - Geometry is typically based on mid-plane geometry
  - Does not include thermal-only relevant items (e.g. TPS, MLI)

In case of perfect matching geometry:

- Non-matching meshes

esa

Mechanical Engineering Department
Thermal and Structures Division

Overview of functionality

- Mapping building functionality
  - Automated mapping for thermal and structural meshes on coinciding surfaces
  - Manual mapping is supported for meshed non-matching/non-coinciding geometry
  - Manual mapping is supported for thermal nodes without related geometry
- User interface
  - Graphical user interface for manipulation of model correspondence fully integrated in MSC.PATRAN (implemented in PCL)
  - Other modules require command-line manipulations

esa

Mechanical Engineering Department
Thermal and Structures Division

Overview of functionality (cont)

- Supported lumped parameter codes: Any
  - temperature have to be converted to SINAS format (converters provided)
- Supported FE formats: MSC.NASTRAN, ASKA
  - Generates FE node temperature loads
  - Generates element temperature loads for cross-sectional thermal gradient (e.g. for honeycomb panels)
  - Requires NASTRAN DMAP license for matrix extraction

Mechanical Engineering Department
Thermal and Structures Division

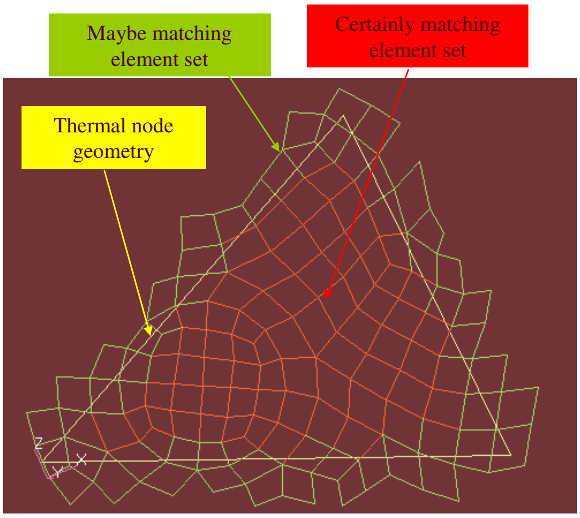
The method in 3 steps

Build Correspondence	MSC.PATRAN GUI
↓	
Matrix generation	Command-line/batch modules in combination with FEM-code
↓	
Solve interpolation equations	Command-line/batch module

Mechanical Engineering Department
Thermal and Structures Division

The method in 3 steps
Step 1: Correspondence building

- Build correspondence/mapping between:
  - Lumped parameter thermal nodes
  - Structural finite elements
- Using MSC.PATRAN GUI
- Automatic overlap detection



Mechanical Engineering Department
Thermal and Structures Division

The method in 3 steps
Step 2: Matrix generation

- Constraint matrix: Prescribed average temperature
  - Weighted average temperature of FE nodes of elements corresponding with thermal node is equal to thermal node temperature
- Element shape functions used to obtain weighting coefficients

$$T_j^t = \sum_i a_i T_i^f \Rightarrow \underline{T}^t = A\underline{T}^f$$

$$\sum_i a_i = 1$$

Constraint matrix

T_i^f = FEM node temperature

T_j^t = Thermal node temperature

a_i = Weighting coefficient

Mechanical Engineering Department
Thermal and Structures Division

The method in 3 steps
Step 2: Matrix generation (cont)

- Conduction matrix for structural FEM-mesh
  - Use structural finite element model
  - Replace structural material properties with thermal conductive material properties
  - Use FEM-code (NASTRAN or ASKA) to extract the conduction matrix

esa

Mechanical Engineering Department
Thermal and Structures Division

The method in 3 steps
Step 3: Solving interpolation equations

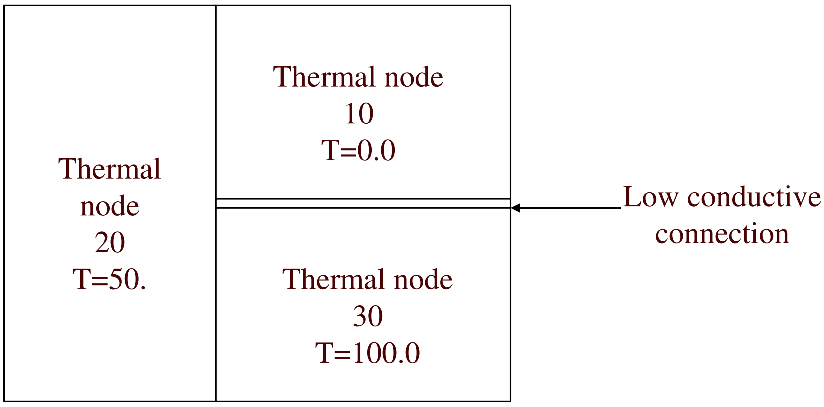
- Solve the interpolation system equations for the FEM-node temperatures $\underline{T}^f$:

$$\begin{bmatrix} C & A^T \\ A & 0 \end{bmatrix} \begin{bmatrix} \underline{T}^f \\ \underline{q} \end{bmatrix} = \begin{bmatrix} \underline{0} \\ \underline{T}^t \end{bmatrix}$$

esa

Mechanical Engineering Department
Thermal and Structures Division

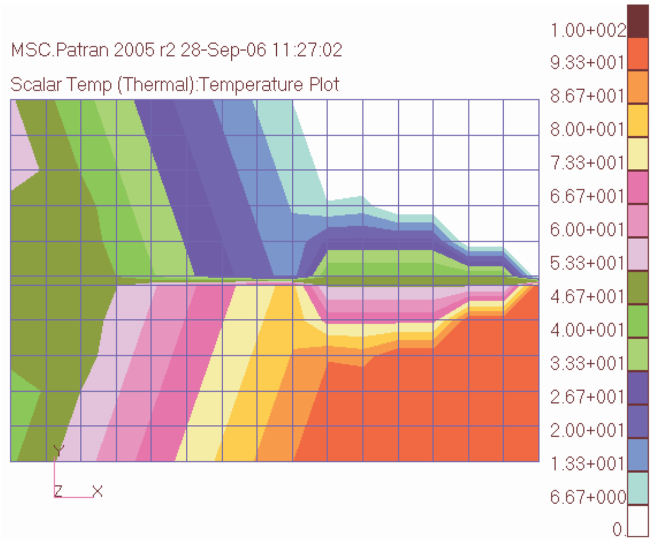
Example



Mechanical Engineering Department
Thermal and Structures Division

Example (cont)

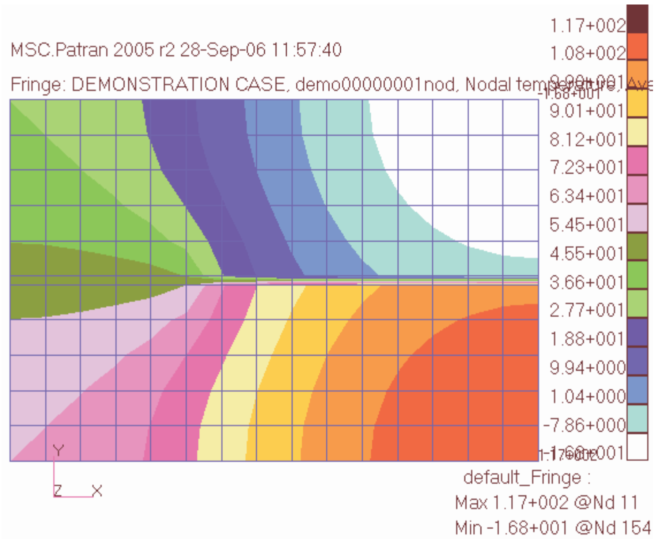
Purely geometrical interpolation of TLP temperatures on FE mesh



Mechanical Engineering Department
Thermal and Structures Division

Example (cont)

SINAS results



esa

Mechanical Engineering Department
Thermal and Structures Division

Projects that used SINAS

- Ariane 5 Engine Frame (Dutchspace)
- GAIA, LISA path finder (ESA ESTEC)

esa

Mechanical Engineering Department
Thermal and Structures Division

How to get SINAS

- The SINAS IV software can be obtained from:

https://exchange.esa.int/restricted/sinas/

- Through the web-page access to the download area can be requested

Mechanical Engineering Department
Thermal and Structures Division