COMPASS
www.compassis.com

# RamSeries - Validation Case 38

## Thermal analysis for shells with heat flow and given temperature

RamSeries

Version
15.1.0

Compass Ingeniería y Sistemas http://www.compassis.com
Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - E: info@compassis.com - C/ Tuset 8, 7o 2a, Barcelona 08006 (Spain)

**http://www.compassis.com**
**info@compassis.com**
**November 2018**

# Table of Contents

# 1 Validation Case 38 - Thermal analysis of a shell with heat flux

## Model Description

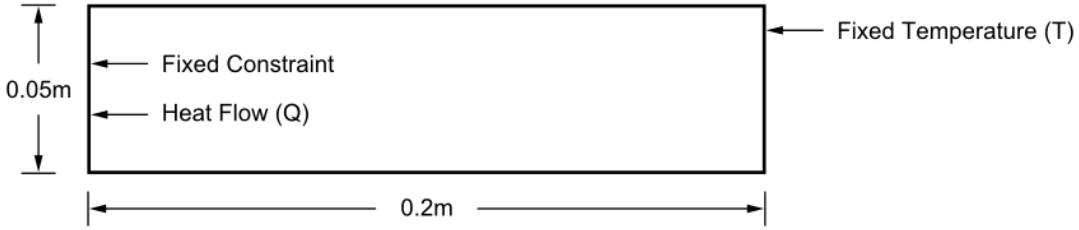
The aim of this test is to check the accuracy of a thermal analysis combined with a structural analysis. In this case, both analysis are performed separately and the structural analysis read temperature distribution in a results file created during thermal analysis calculation.

A shell is fixed along the width on one edge and heat flow is applied on the same edge. The oppossite edge is subjected to a temperature of 20ºC. Ambient temperature is 20ºC.

In this validation case, maximum temperature, maximum total heat flux and maximum total deformation are compared with results shown in Reference[1].

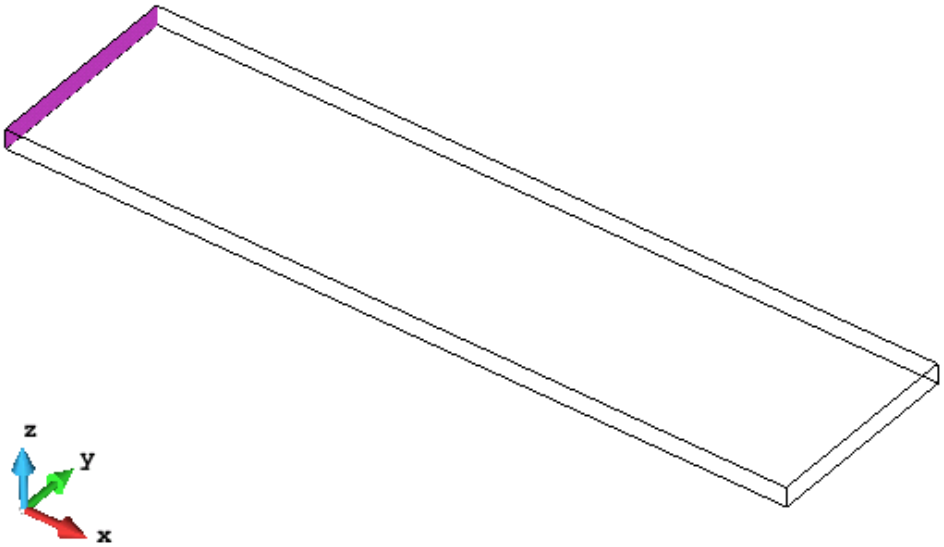
### Geometric model

Following, a schematic picture is shown with dimensions.



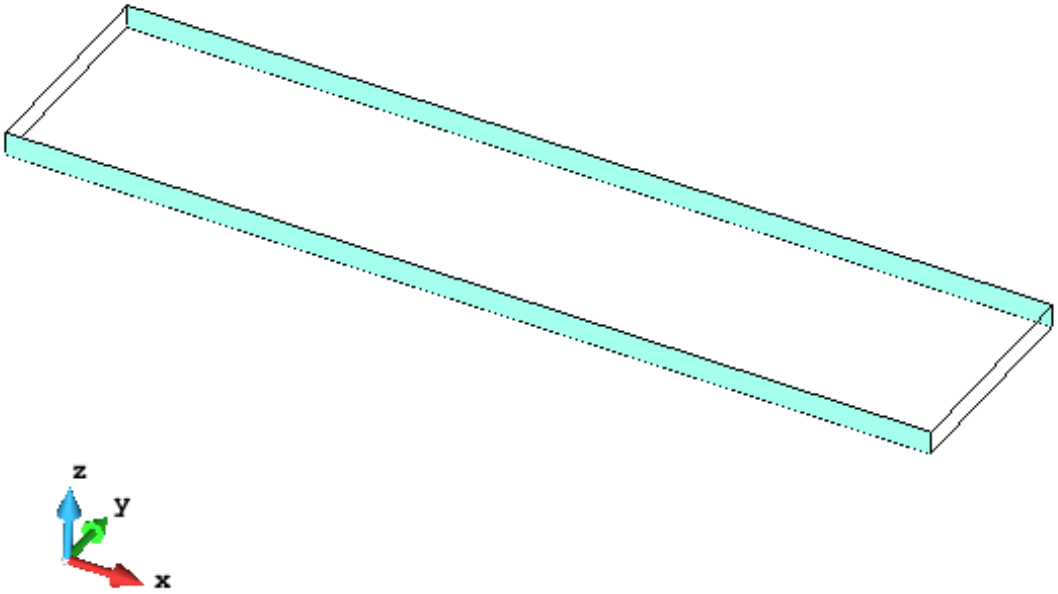

Thickness = 0.005 m; Q = 5 W; T= 20ºC

Since TDYN only allows to perform thermal analysis with solid elements, then the shell is modelled as solid. The next picture shows the model used in thermal and structural analyses.

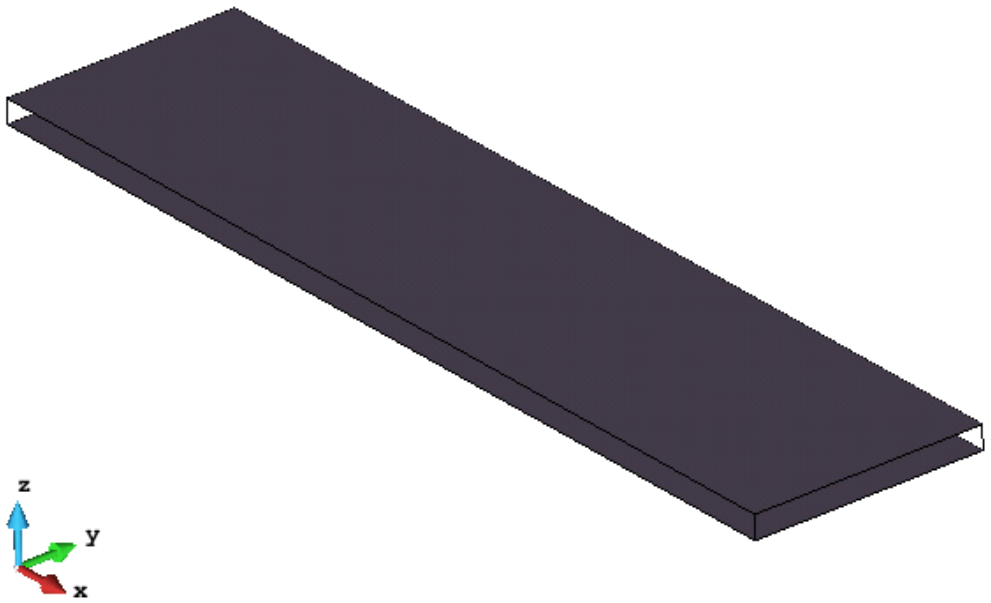



The colored face is fixed constrained and a heat flux of 5 W is applied. The opposite face is subjected to a temperature of 20 ºC.

To ensure the displacement along a single axis, the next constraints must be applied.




Constraint in Y direction




Constraint in Z direction

### Temperature conditions

- Heat flow:
  - Q = 5 W
  - Face area = 2.5e-4 $m^2$
  - Heat flux = 20000 $W/m^2$
- Fixed temperature = 20 ºC
- Ambient temperature = 20 ºC

### Material properties

E = 2e11 Pa (Young's modulus)

v = 0.0 (Poisson's ratio)

α = 1.2e-5 1/ºC (thermal expansion coefficient)

k = 60.5 W/m·ºC (thermal conductivity)

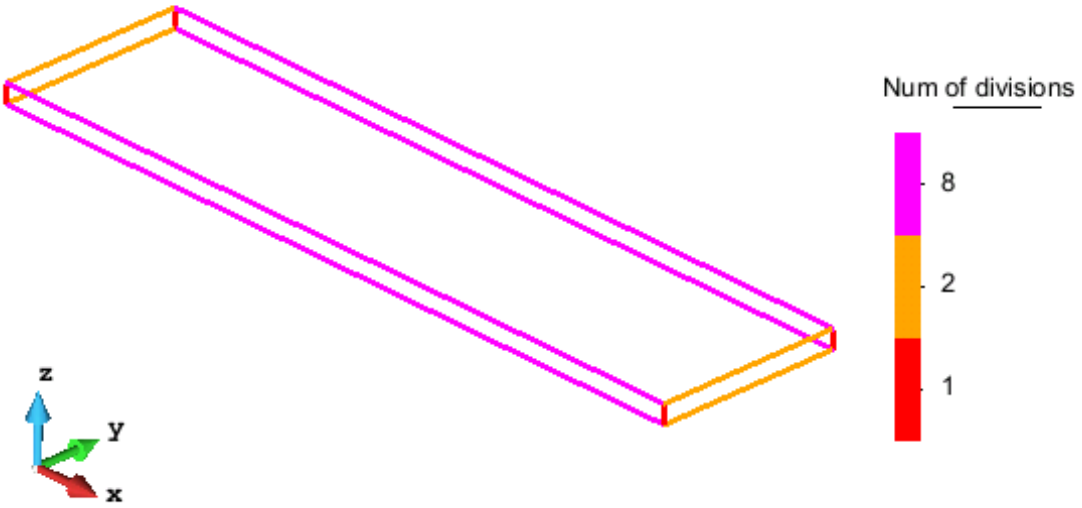
## Results

A thermomechanic problem must be solved in two steps:

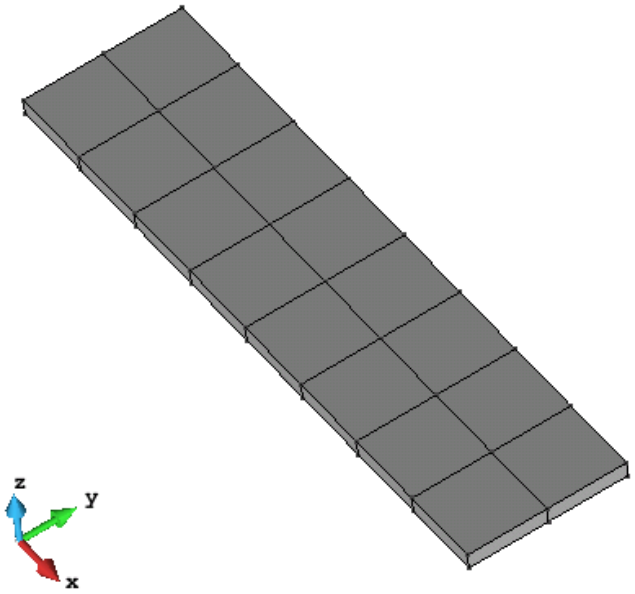
- A thermal analysis allows to know the temperature distribution and the heat flux.
- A mechanic analysis that calculates the displacement using the results obtained in the previous thermal analysis.

### Mesh

A structured mesh of hexaedra is used in both analysis. The number of divisions done in each edge is shown in the following picture and the mesh too:



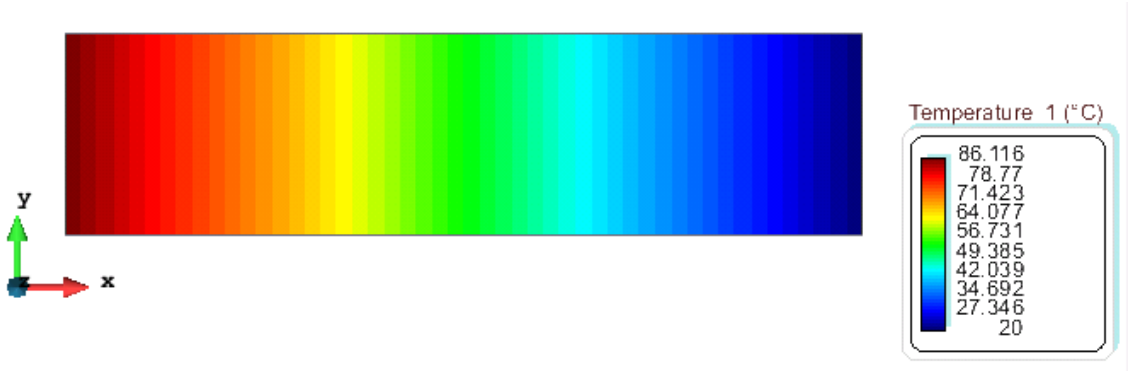

Number of divisions in each edge




Mesh of hexaedra. This mesh contains 54 nodes and 16 elements

## Thermal analysis results

The following pictures show the temperature and heat flux distribution.

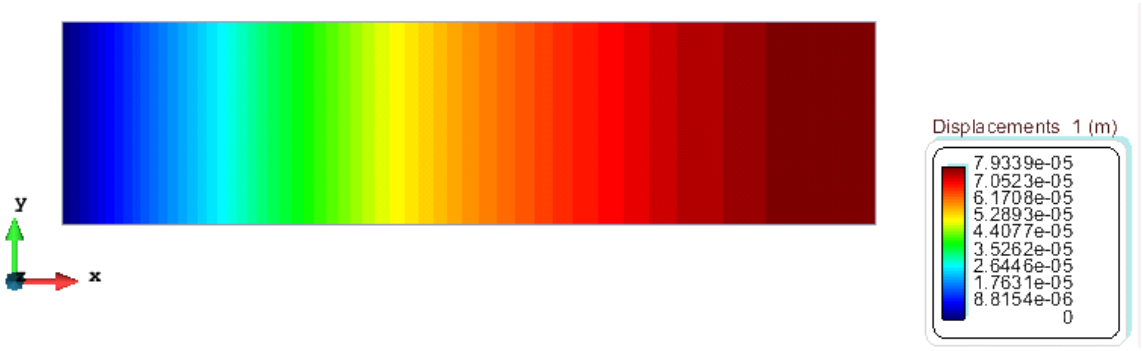



Temperature distribution. Maximum temperature = 86.116 ºC


Heat flux distribution. Heat flux = 20000 W/m2

## Structural analysis

And finally, the displacement results are shown in the next picture.




Displacement distribution. Maximum displacement = 7.9339e-5 m

## Summary of results

The next table shows the results obtained by TDYN and RAMSERIES and the results in the Reference[1].

| | Target | Numerical result | Error [%] |
|---|---|---|---|
| Maximum temperature [ºC] | 86.116 | 86.116 | 0.0 |
| Maximum total heat flux [$W/m^2$] | 2e4 | 2e4 | 0.0 |
| Maximum displacement [m] | 7.9339e-5 | 7.9339e-5 | 0.0 |

## References

[1] ANSYS. Verification Manual for Workbench, pag 65-66. April 2009. Release 12.0.

## Validation Summary

| | |
|---|---|
| CompassFEM version | 15.1.0 |
| Tdyn solver version | 15.1.0 |
| RamSeries solver version | 15.1.0 |
| Benchmark status | Successfull |
| Last validation date | 27/11/2018 |