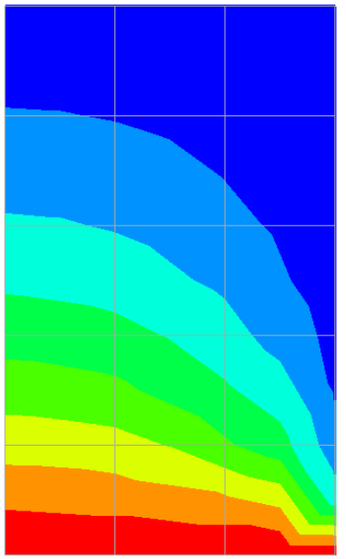
Tutorial

Two-Dimensional Heat Transfer with Convection



DIANA

DIANA

Outline

1 Description 3

2 Finite Element Model 4
- 2.1 Geometry . . . 7
- 2.2 Properties . . . 8
- 2.3 Boundary Conditions . . . 9
  - 2.3.1 Convection Edges . . . 9
  - 2.3.2 Bottom Edge . . . 11
- 2.4 Mesh . . . 12

3 Analysis 14
- 3.1 Commands . . . 14
- 3.2 Results . . . 15
  - 3.2.1 Temperature Distribution . . . 15
  - 3.2.2 Heat flux . . . 17

Appendix A Additional Information 18

1 Description

This example is based on the NAFEMS benchmark T4[1]. The model is a 0.60 by 1.00 m rectangle $ABCD$ with uniform thickness [Fig. 1]. Edge AB has a prescribed temperature of 100 °C. Edge AD is completely insulated. There is convection to an ambient temperature of 0 °C along edges BC and CD with a surface conduction coefficient K = 750 W/(m^2.°C). There is no internal heat generation. The conductivity k of the material is 52 W/(m.°C). The target value for the temperature at point E is 18.3 °C.

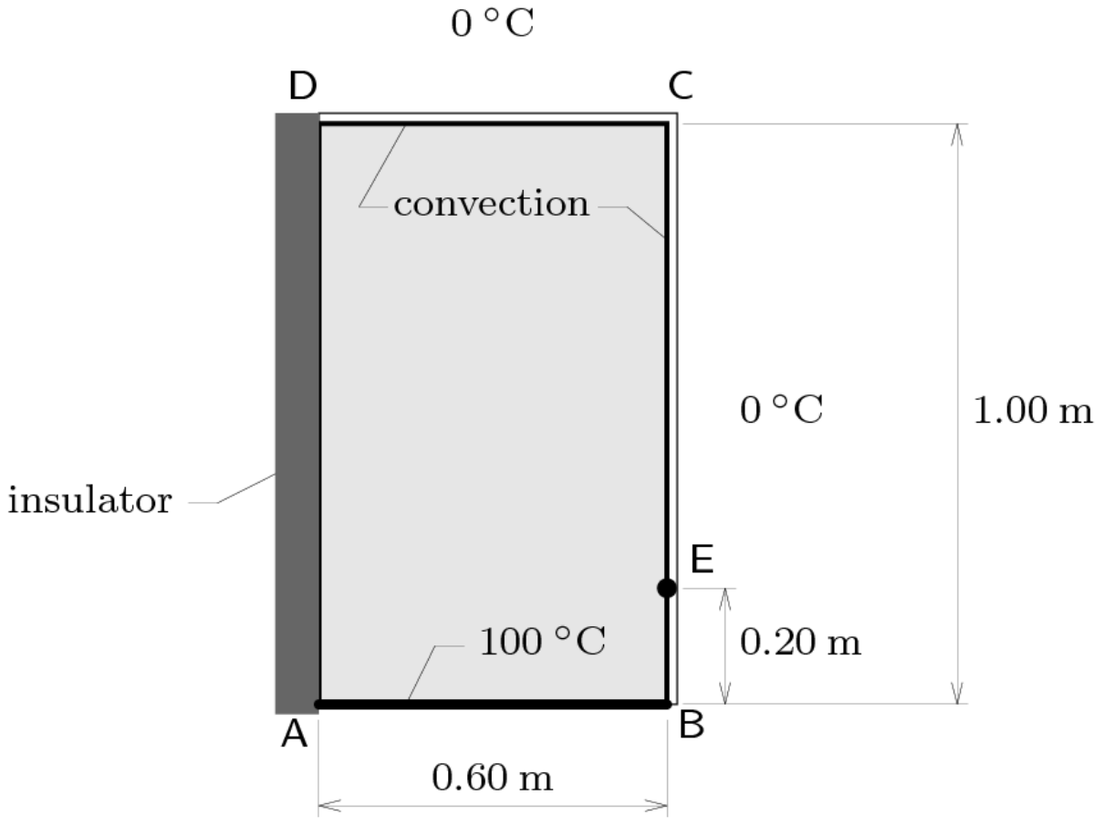


Figure 1: Two-dimensional heat transfer with convection

[1]NAFEMS, *The Standard NAFEMS Benchmarks*, 1990

2 Finite Element Model

We start a new project for a two-dimensional heat flow finite element model.

DIANAIE

Main menu → File → New [Fig. 2]

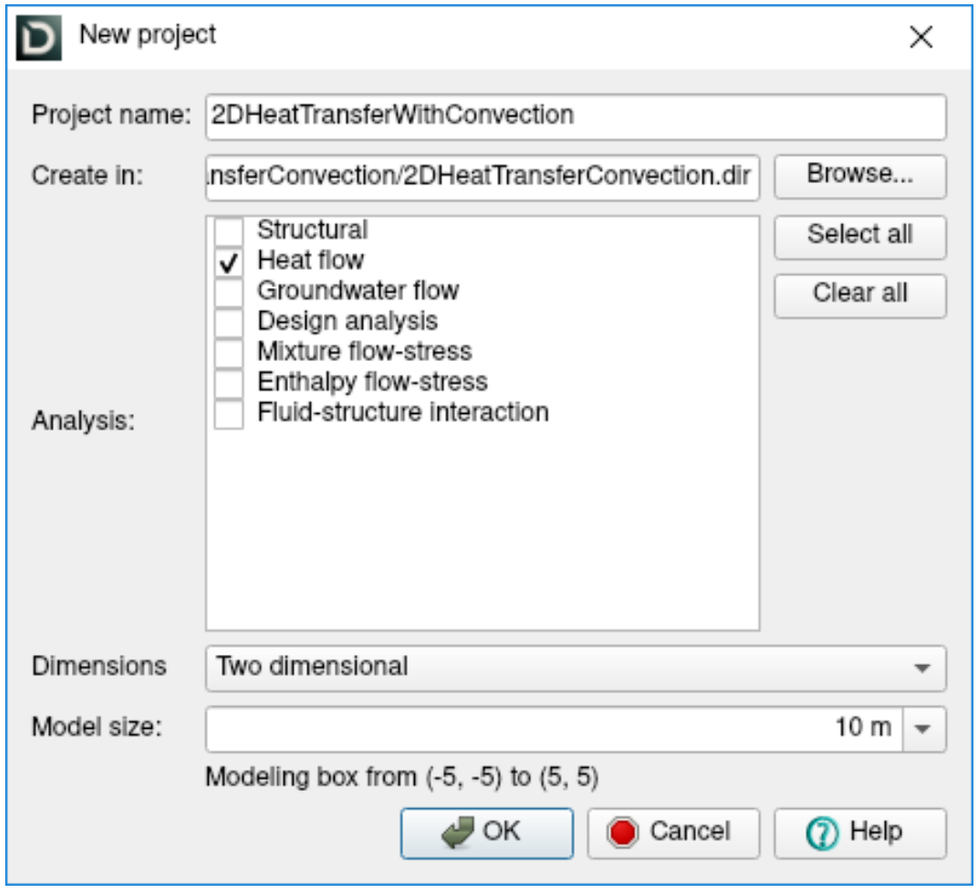


Figure 2: New project dialog

We choose millimeter for the quantity length, ton for mass and degree Celsius for temperature.

Geometry browser → Reference system → Units [Fig. 3]
Property Panel [Fig. 4]

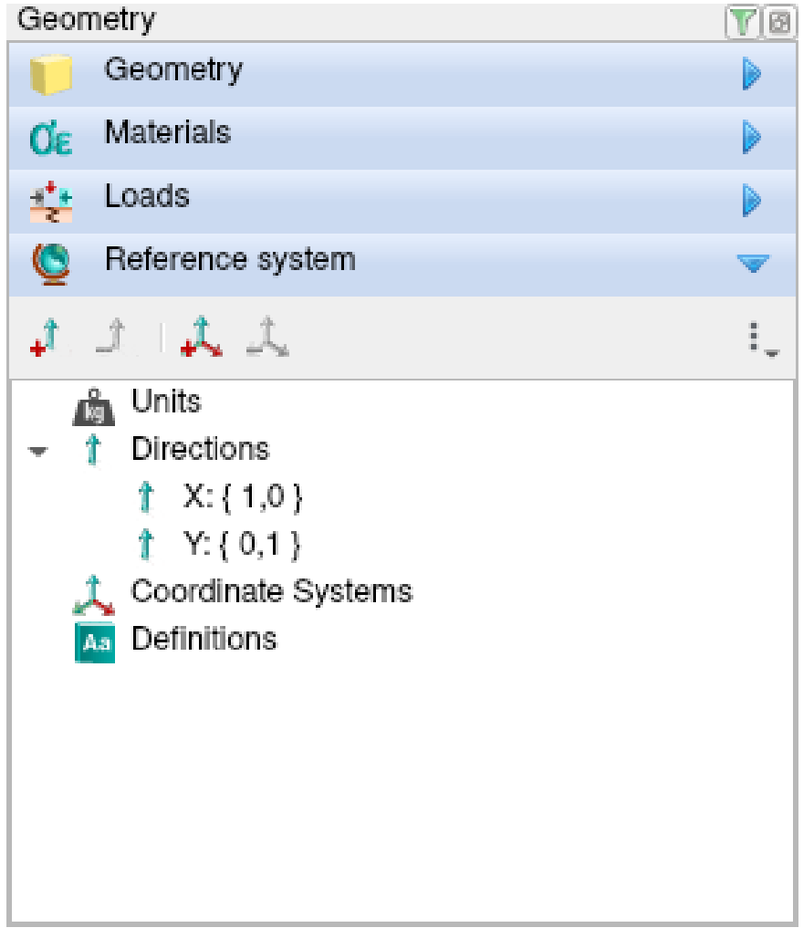


Figure 3: Geometry browser

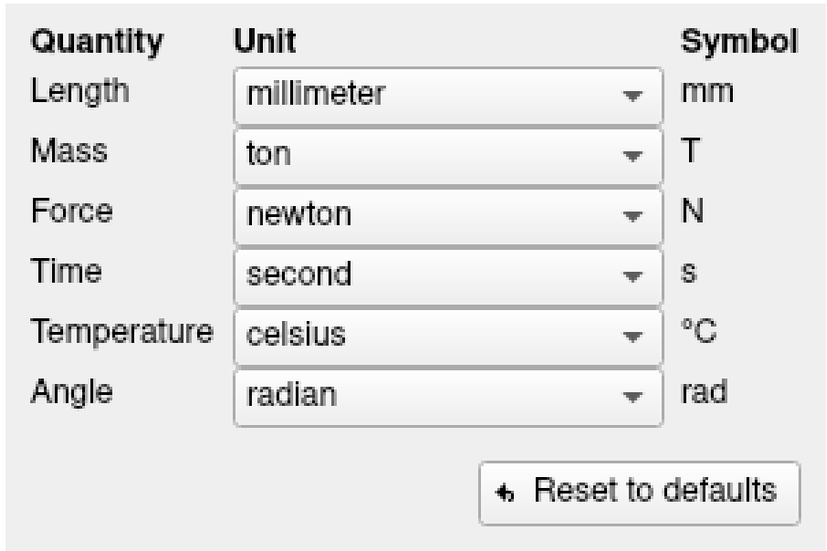


Figure 4: Property panel - units

For the mesh preferences we choose hexagonal/quadrilateral quadratic elements.

Main menu → Mesh → Mesher preferences [Fig. 5]

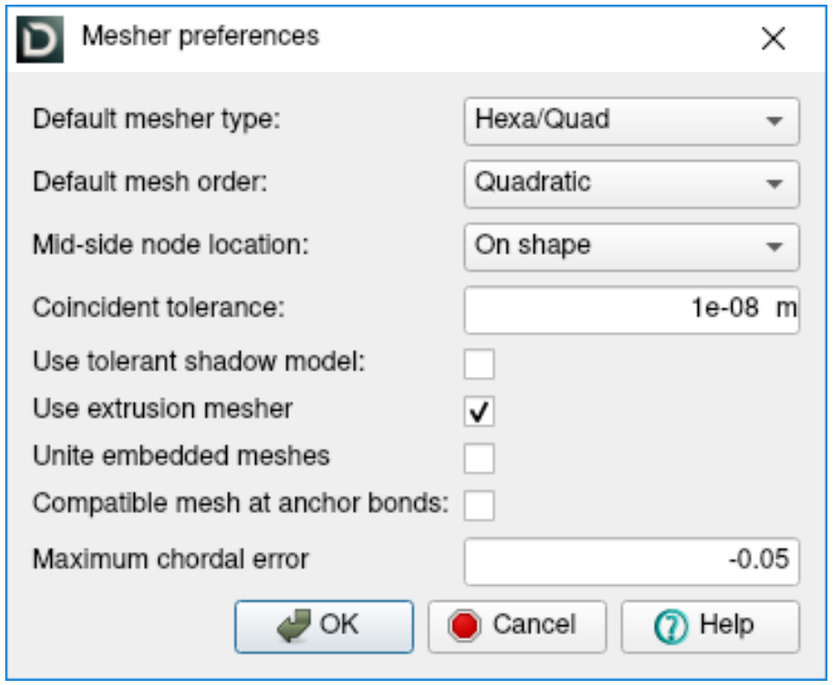


Figure 5: Mesher preferences

2.1 Geometry

For the geometry a polygon sheet is defined. The coordinates are tabulated in Figure 6.

DIANAIE

Main menu → Geometry → Create → Polygon sheet [Fig. 6]

Main menu → Viewer → Fit all [Fig. 7]

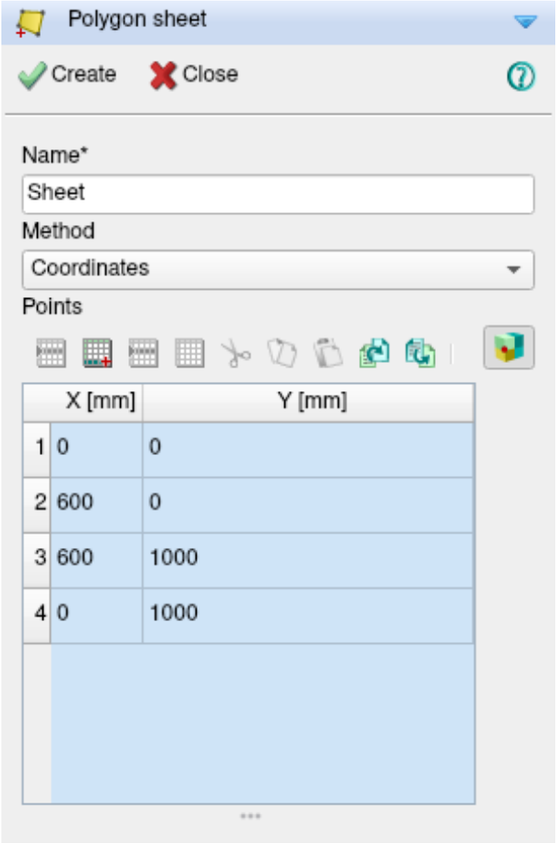


Figure 6: Add polygon sheet

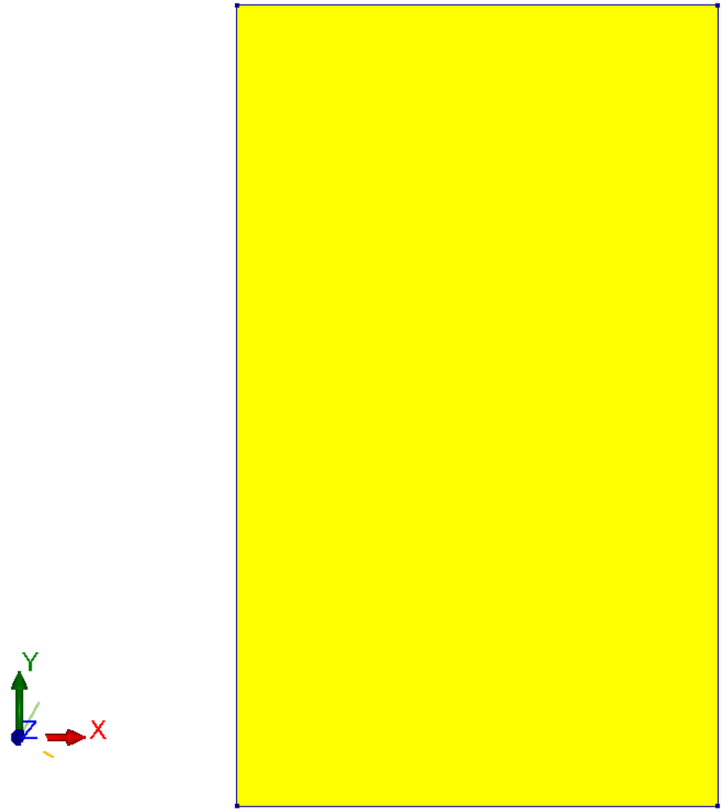


Figure 7: View of model - *Sheet*

2.2 Properties

We assign the element class and the material properties to the single polygon sheet. We use two-dimensional flow elements. An isotropic heat flow material model is defined with a the thermal conductivity of 52 W/(m.℃) = 52 T.mm/(s^3.℃). The thermal capacity is null.

DIANAIE

Main menu → Geometry → Assign → Shape properties [Fig. 8]
Shape properties → Material → Add material [Fig. 9] → Edit material [Fig. 10]

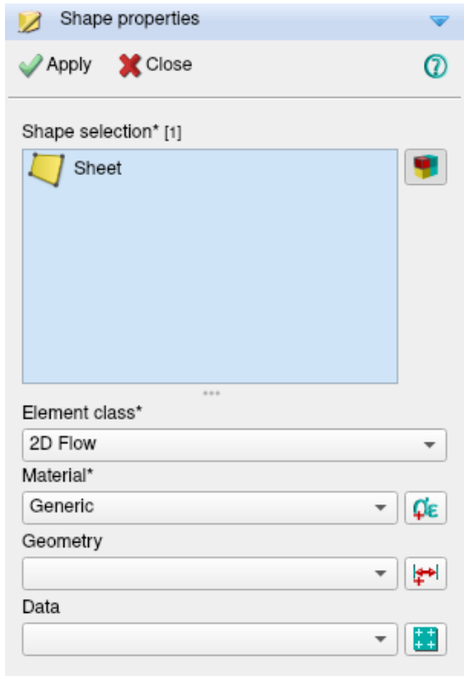


Figure 8: Property assignment

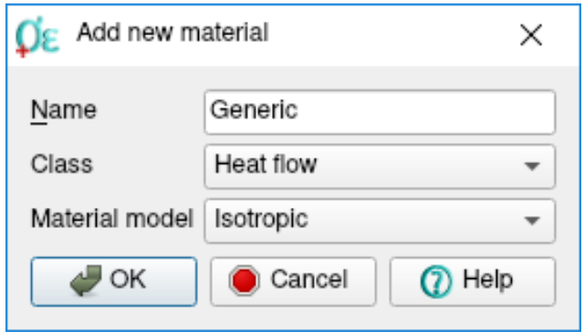


Figure 9: Add new material

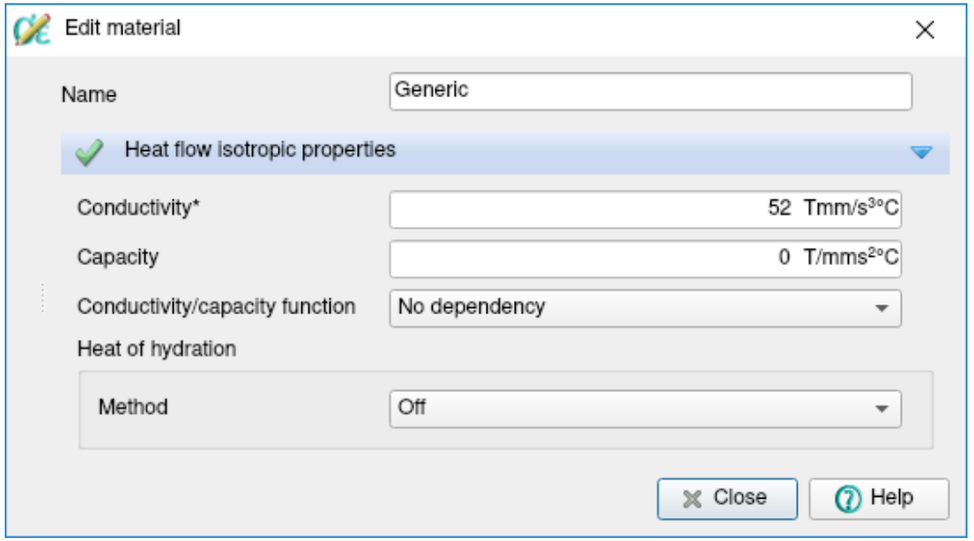


Figure 10: Edit material

2.3 Boundary Conditions

2.3.1 Convection Edges

An ambient temperature of 0 °C is set at the top and right edge where heat transport is taking place in the form of convection; this set of edges is therefore identified as *convection edges*. In order to assign this external temperature (boundary condition) a boundary interface is defined along the aforementioned edges (see Figure 11 and Figure 14) which is treated as a material in order to assign thermal properties to these edges. The interface material is named *Convection* [Fig. 12].

DIANAIE

Main menu → Geometry → Assign → Connections [Fig. 11]
Edit connections → Material → Add material [Fig. 12] → Edit material [Fig. 13]

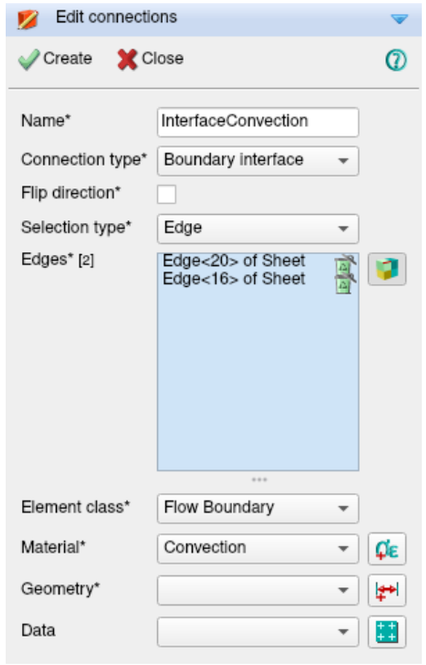


Figure 11: Boundary interface at convection edges

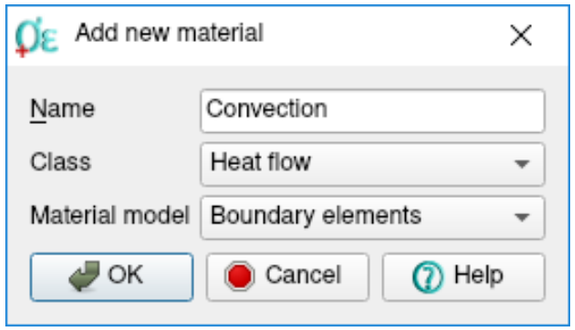


Figure 12: Add new material

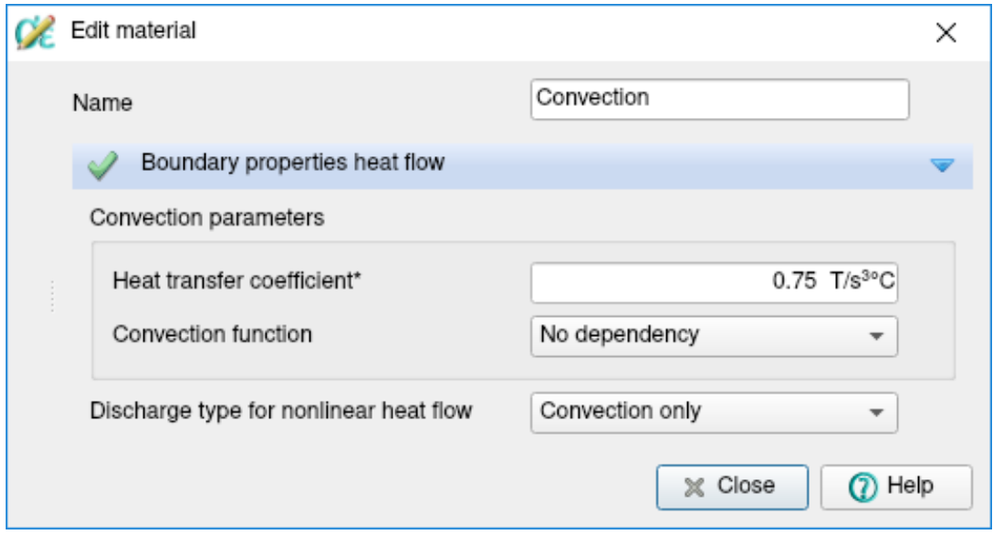


Figure 13: Edit material

Main menu → Geometry → Assign → Thermal boundary conditions [Fig. 14]

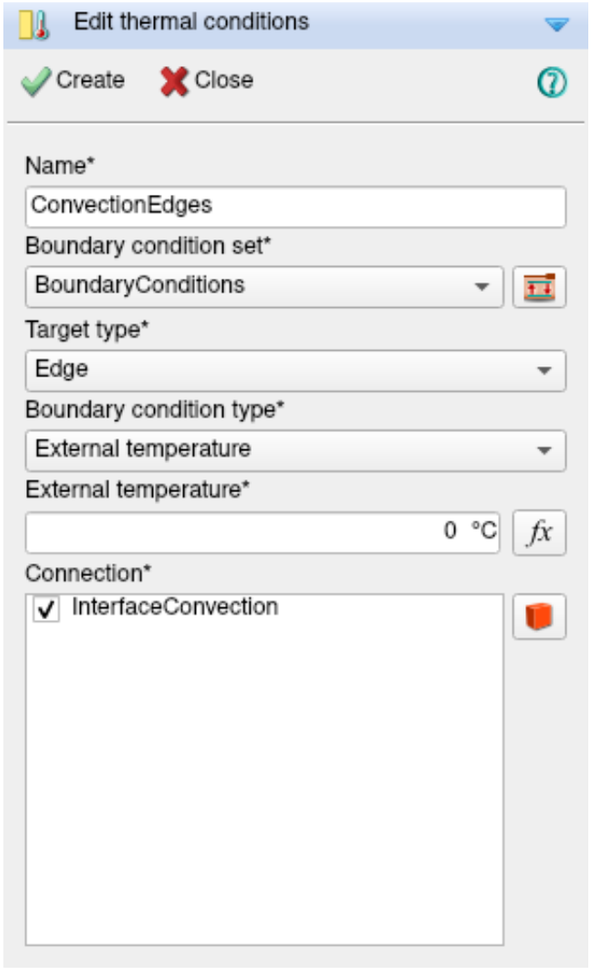


Figure 14: Ambient temperature at convection edges

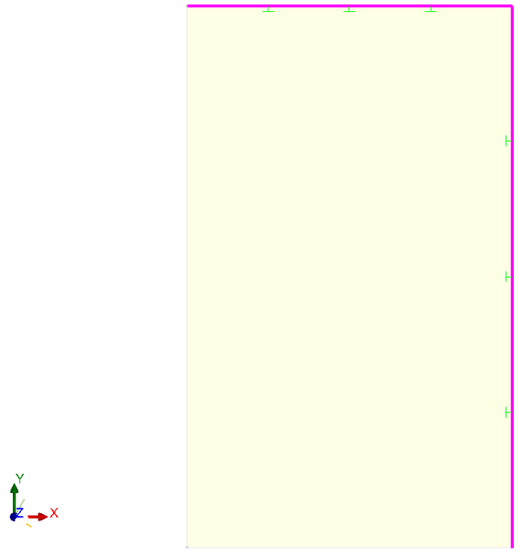


Figure 15: Convection edges

2.3.2 Bottom Edge

We define a thermal boundary condition with a prescribed temperature of 100 ℃ at the bottom edge. We use automatic option for the creation of the corresponding fixed temperature.

DIANAIE

Main menu → Geometry → Assign → Thermal boundary conditions [Fig. 16]

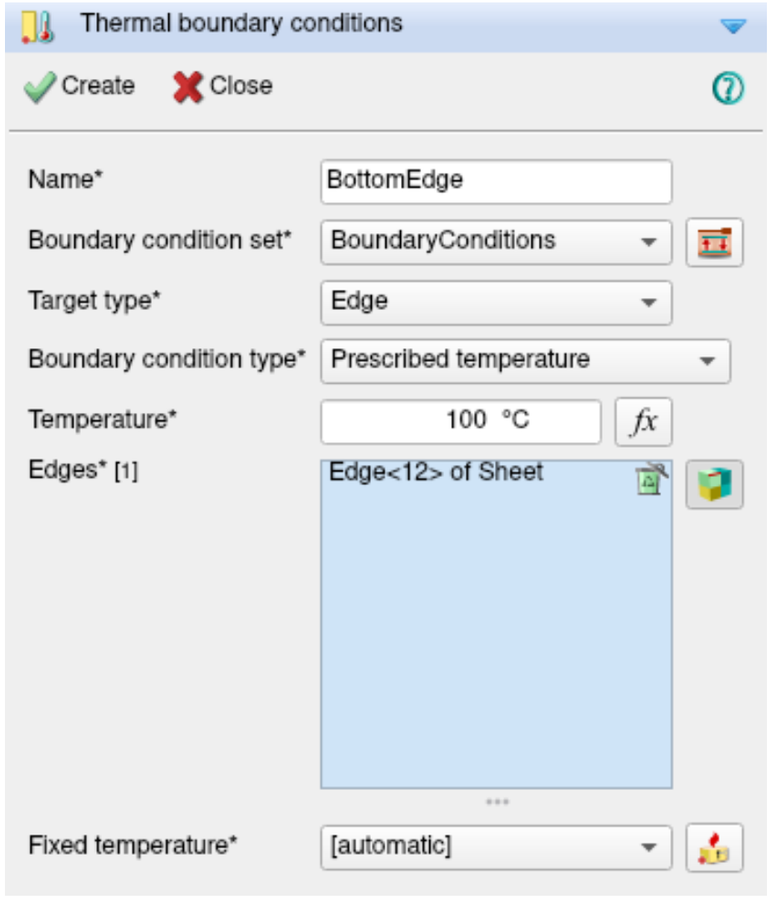


Figure 16: Fixed temperature specified

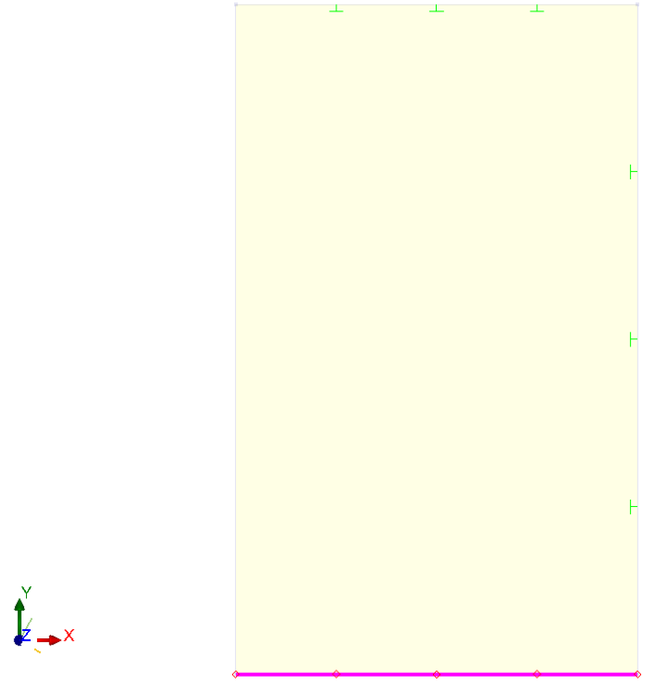


Figure 17: Bottom edge

2.4 Mesh

The two-dimensional finite element model consists of fifteen eight-noded quadrilateral flow elements and along edges BC and CD three-noded boundary flow elements are applied; five and three respectively. The mesh is constructed by dividing the bottom and top edge in three equally sized line segments [Fig. 18] and the left and right edge in five equally sized line segments [Fig. 20]. With this, the geometry is subdivided into fifteen quadrilaterals which is a requirement for this benchmark.

DIANAIE

Main menu → Geometry → Mesh → Mesh properties [Fig. 18] [Fig. 20]

Main menu → Geometry → Mesh → Generate mesh

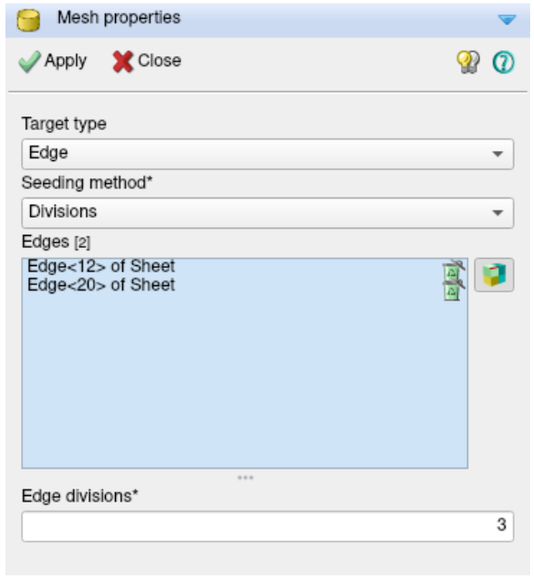


Figure 18: Mesh properties for bottom and top edges

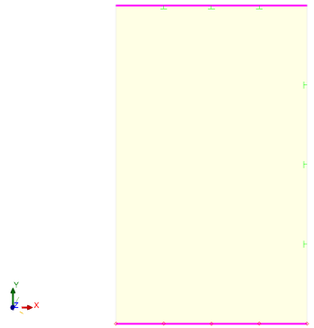


Figure 19: Bottom and top edges

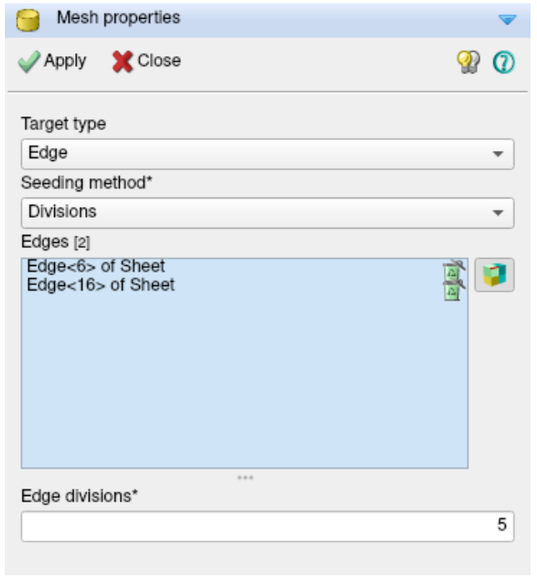


Figure 20: Mesh properties for left and right edges

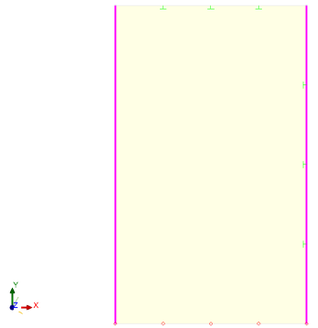


Figure 21: Left and right edges

To find out whether or not the target value for the temperature at point E is reached, we need to know the corresponding node number (in this case it is number 11 [Fig. 22])[2].

DIANAIE

Main menu → Mesh → Mesh → Shrunken shading

Main menu → Viewer → Selection mode → Node selection

Graphics window → Select all → Show ids [Fig. 22]

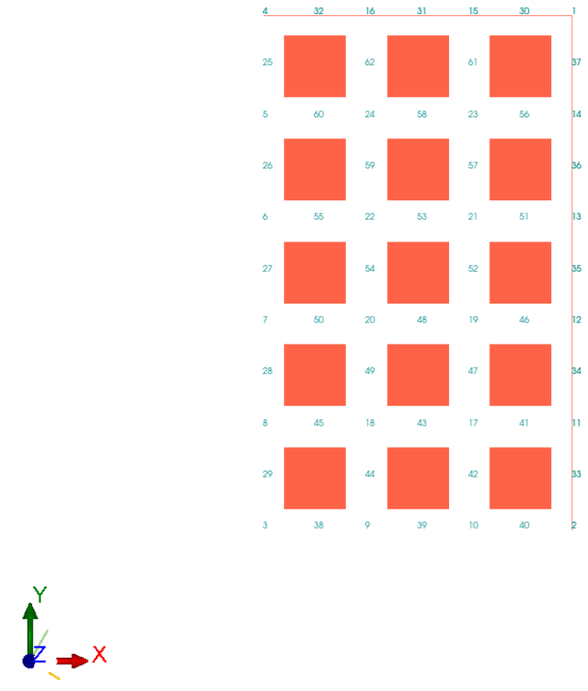


Figure 22: Mesh - shrunken elements and node identification

[2]Note: the node number might change by following different sequences of modeling on mesh generation.

3 Analysis

3.1 Commands

We perform a *Steady state heat transfer* analysis.

DIANAIE

Main menu → Analysis → Add analysis

Analysis browser → Analysis1 → Rename → LinSteStaAn [Fig. 23]

Analysis browser → LinSteStaAn → Add command → Steady state heat transfer [Fig. 24] [Fig. 25]

Main menu → Analysis → Run all analyses

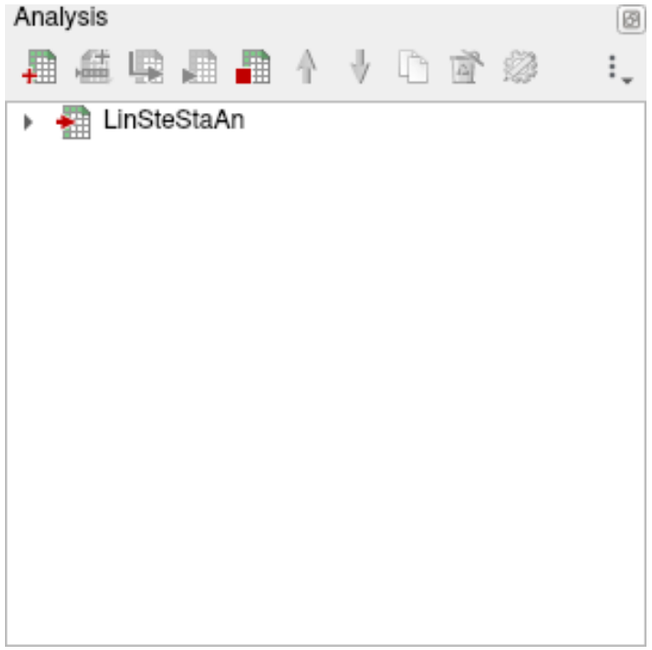


Figure 23: Analysis browser

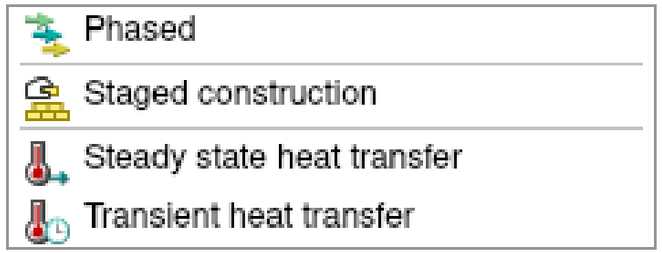


Figure 24: Command menu

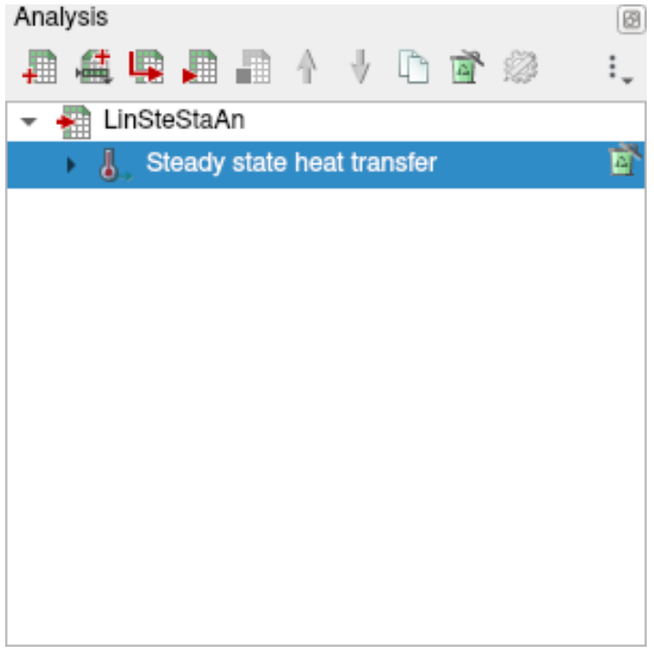


Figure 25: Analysis browser

3.2 Results

3.2.1 Temperature Distribution

The potential temperature distribution PTE is presented [Fig. 27].

DIANAIE

Results browser → LinSteStaAn → Output heat transfer analysis → Nodal results → Temperatures → PTE → Show contours [Fig. 26]

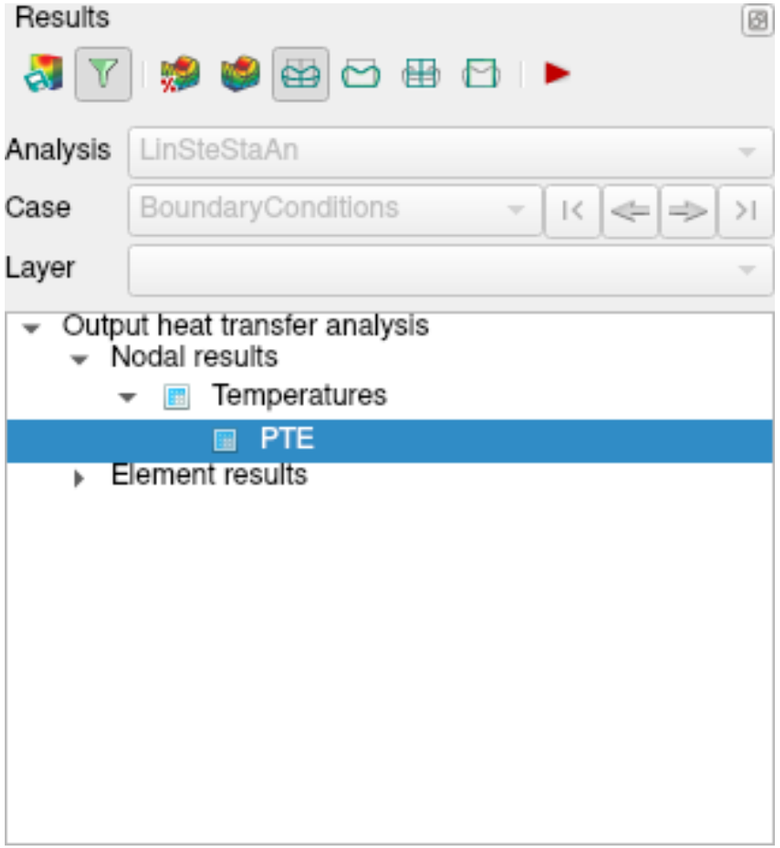


Figure 26: Results browser

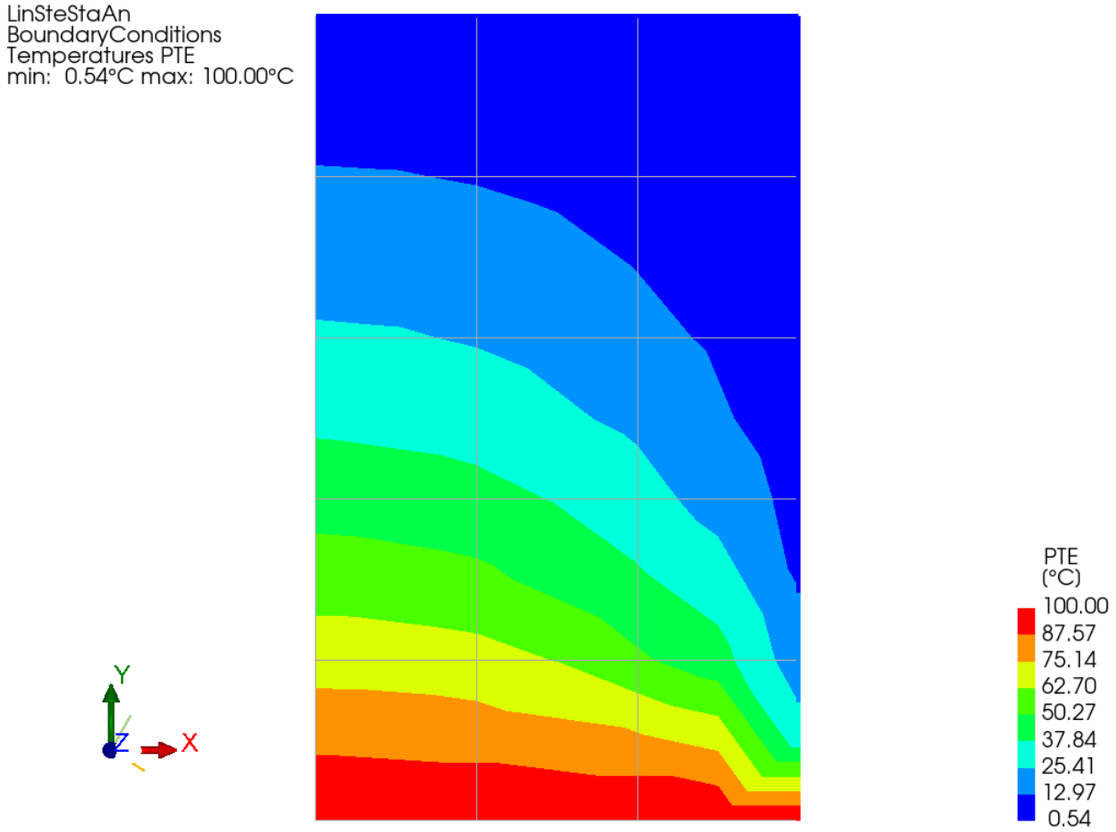


Figure 27: Potential temperature distribution

Figure 29 shows that the temperature at point E (i.e. node 11) is indeed 18.3 °C; neglecting the 0.0022 °C overestimation.

Results browser → LinSteStaAn → Output heat transfer analysis → Nodal results → Temperatures → PTE → Show table [Fig. 29]

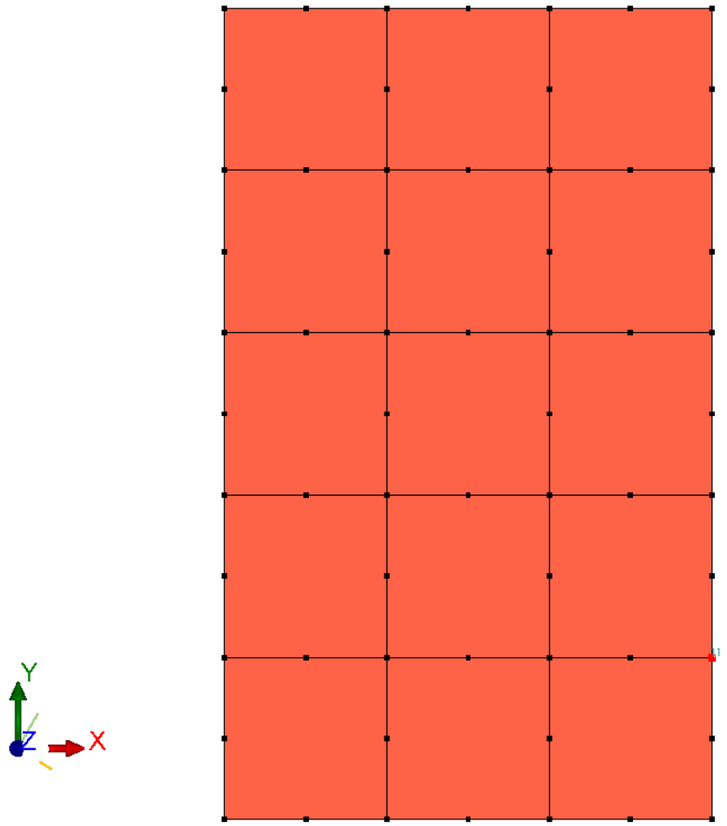


Figure 28: Node selection

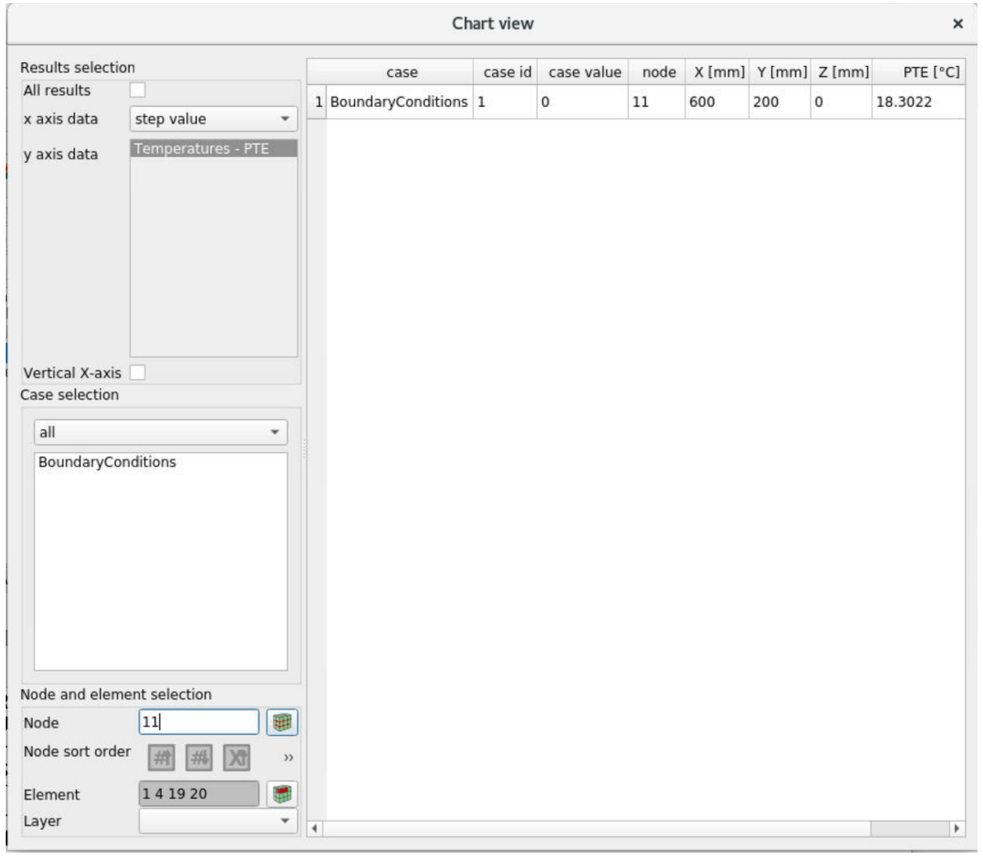


Figure 29: Temperature at point *E*

3.2.2 Heat flux

How the heat flows through the material body can be shown by the heat flux [Fig. 32]. To make it more visible, we scale up the vector plot by increasing the reference value to 3 [Fig. 31]. As expected, the vectors are directed from the bottom edge towards the top and right edges. By not assigning any boundary condition to the left edge, DIANA considers this edge to be fully insulated.

DIANAIE

Results browser → LinSteStaAn → Output heat transfer analysis → Element results → Heat Flux → FLXY → Show vectors [Fig. 30]

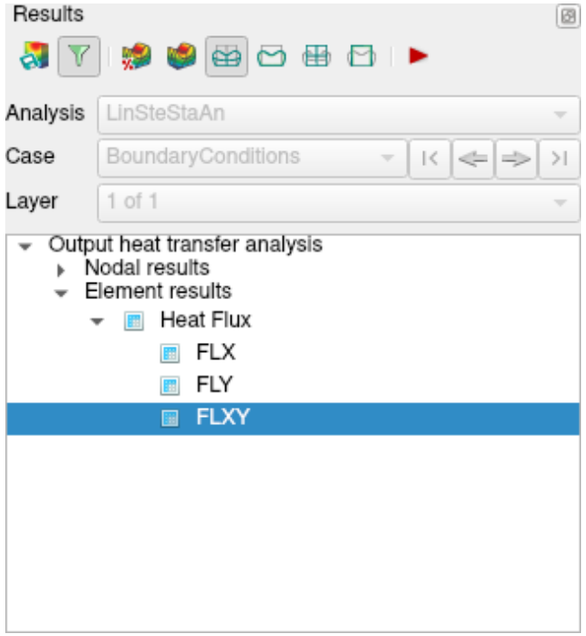


Figure 30: Results browser

Description	Value
Common	
Geometry	
Mesh	
Result	
Deformation settings	
Contour plot settings	
Diagram plot settings	
Vector plot settings	
Scaling factor	1
Vector scaling mode	**Specified value**
Reference value	**3**
Contour levels	Equidistant levels
Equidistant levels options	
Color scale type	Discrete color scale
Color scale limits	Auto-scale to displayed results
Bounding colors	
Background shading	☐
Arrow orientation	Head at node
Tensor plot settings	
Crack plot settings	
Probing curve settings	
Nodal averaging of element results	No averaging

Figure 31: Vector results properties

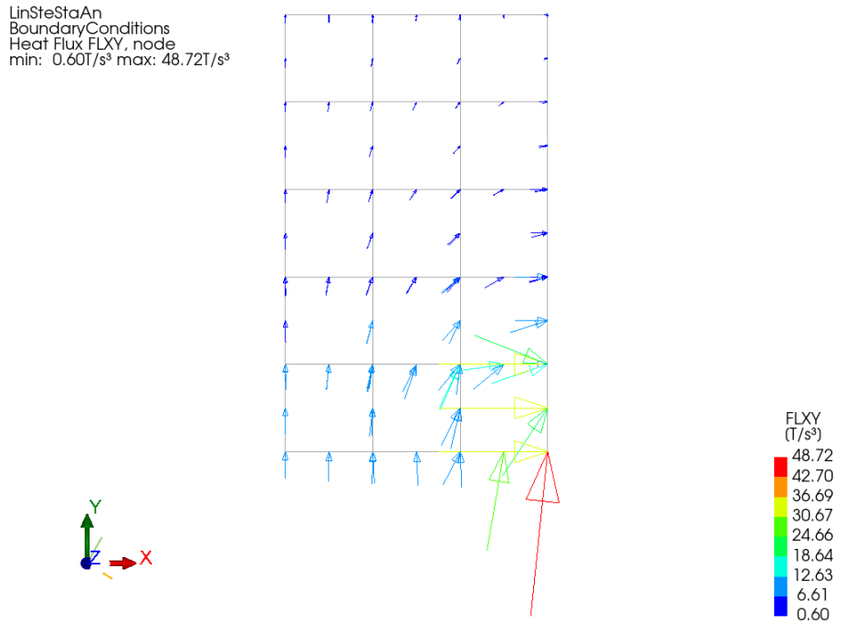


Figure 32: Heat flux through sheet

Appendix A Additional Information

Folder: Tutorials/2DHeatTransferConvection

Number of elements ≈ 15

Keywords:
ANALYS: flow heat steady.
CONSTR: temper.
ELEMEN: bc3ht cq8ht flow potent.
LOAD: elemen node temper.
MATERI: conduc isotro.
OPTION: direct units.
POST: binary ndiana.
PRE: dianai.
RESULT: flux temper total.

References:

[1] NAFEMS. The Standard NAFEMS Benchmarks. Technical report, National Agency for Finite Element Methods and Standards (NAFEMS), 1990.

Laan van Waalhaven 462
2497 GR, The Hague
The Netherlands

DIANAFEA.COM INFO@DIANAFEA.COM +31(0)88 34 262 00

DIANA