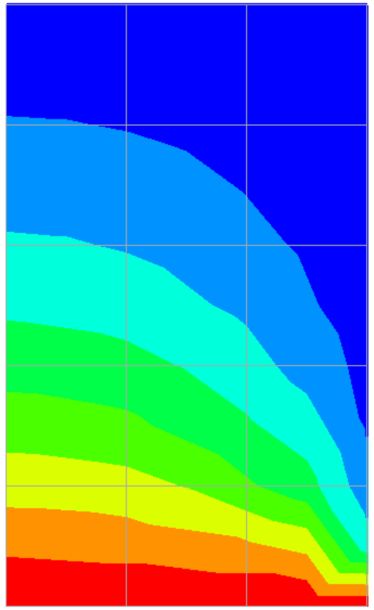

2D Heat Transfer with Convection

Outline

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$^2 \cdot$ °C). There is no internal heat generation. The conductivity k of the material is 52 W/(m $\cdot$ °C). The target value for the temperature at point E is 18.3 °C.

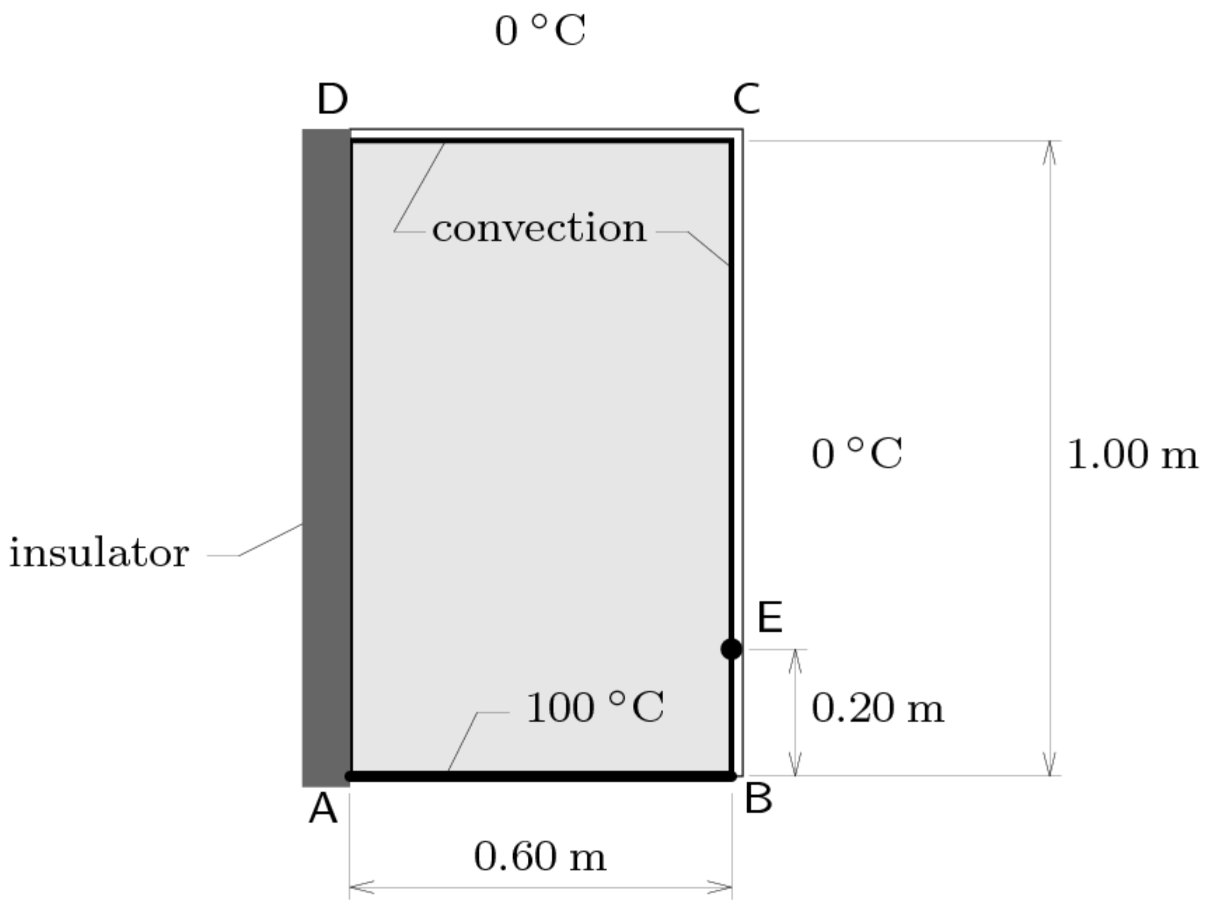

Figure 1: Two-dimensional heat transfer with convection

[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. We choose hexagonal/quadrilateral quadratic elements.

DianaIE

Main menu → File → New [Fig. 2]

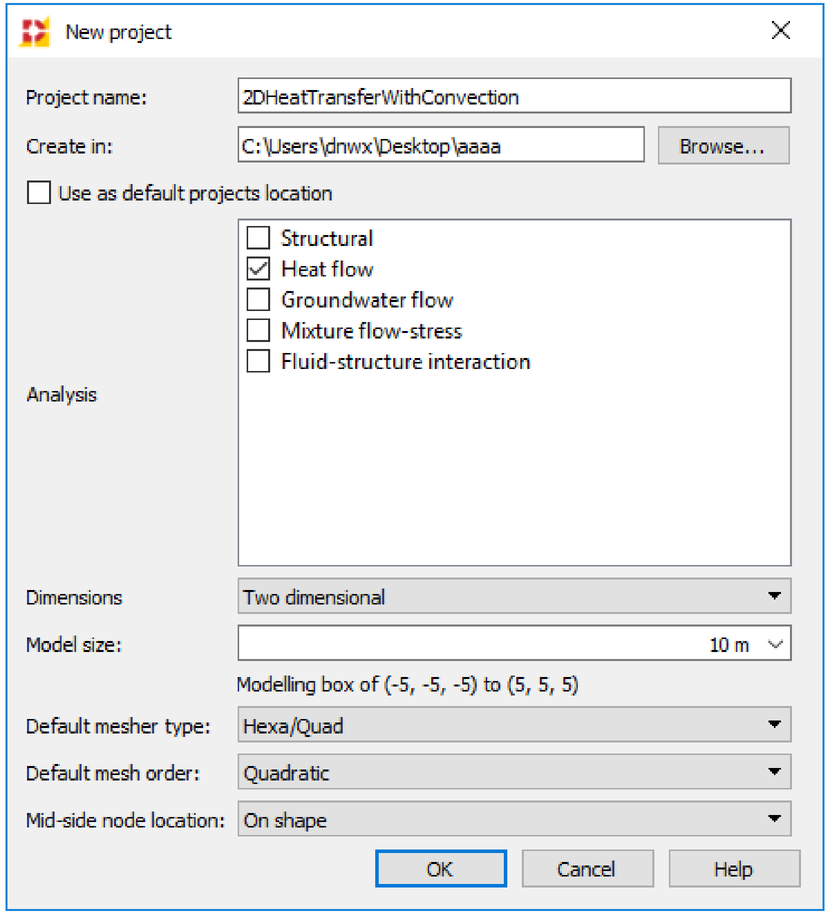

Figure 2: New project dialog

2.1 Units

We choose *millimeter* for the quantity Length, *ton* for Mass and *degree Celsius* for temperature.

DianaIE

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

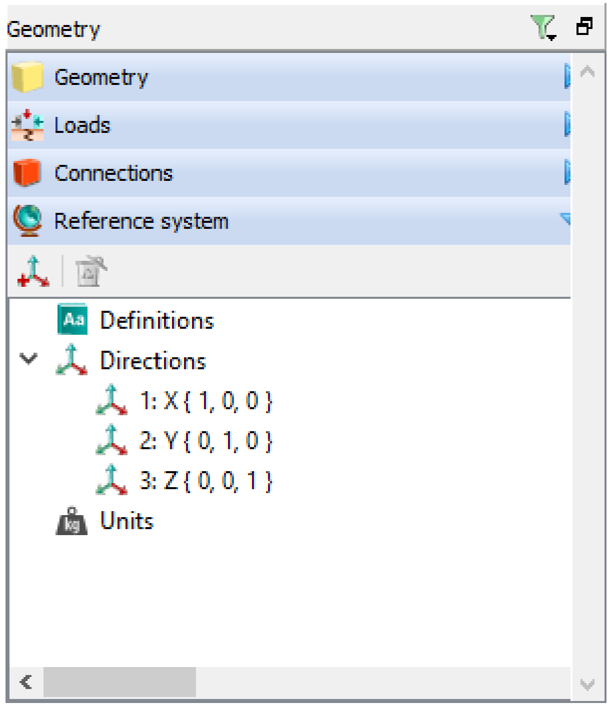

Figure 3: Model windows

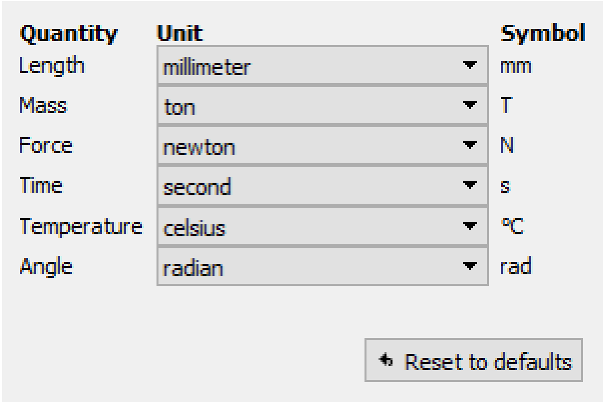

Figure 4: Property Panel - Units

2.2 Geometry Definition

For the geometry a polygon sheet is defined. The coordinates are tabulated in Figure 5.

DianaIE

Main menu → Geometry → Create → Add polygon sheet [Fig. 5]
Main menu → Viewer → Fit all [Fig. 6]

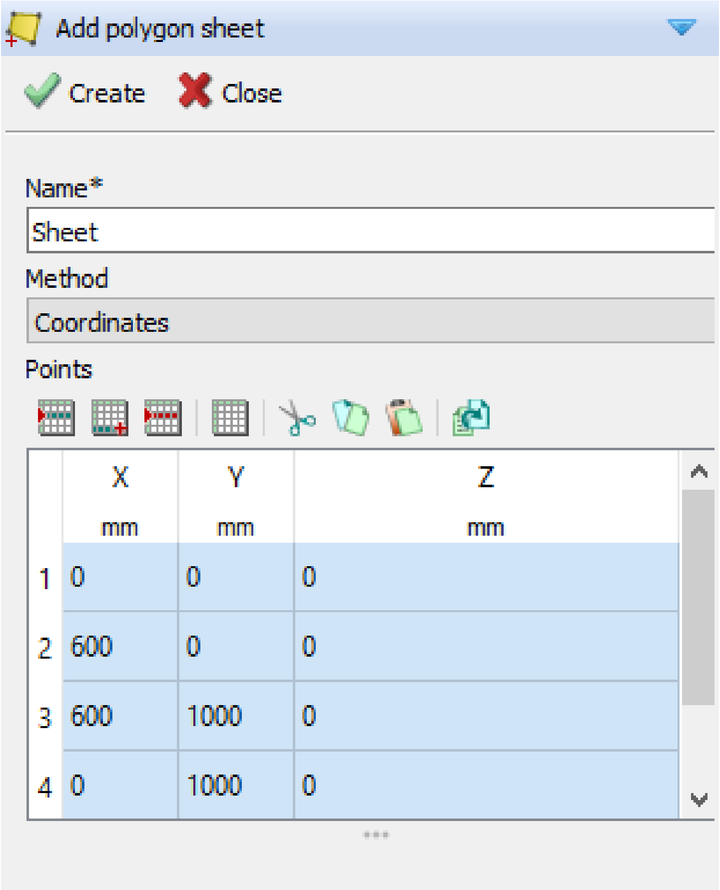

Figure 5: Add polygon sheet

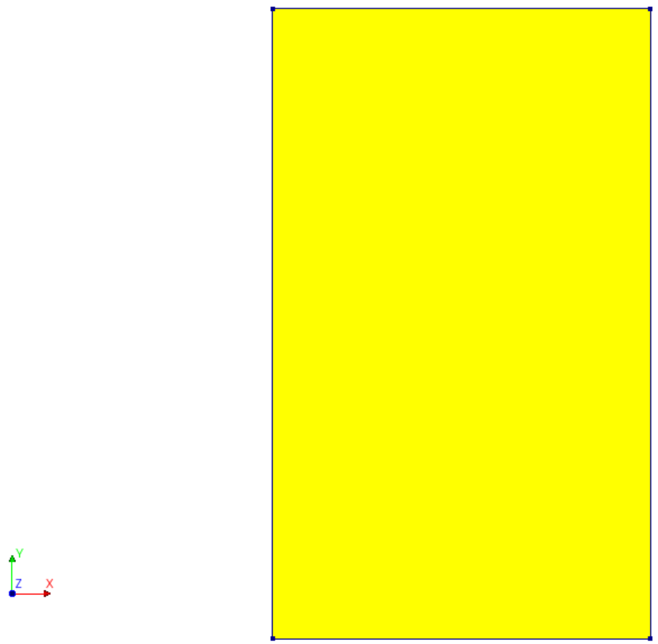

Figure 6: View of model - Sheet

2.3 Properties

We assign the element class and the material properties to the single polygon sheet. Two-dimensional flow elements are used and although not used in the heat transfer analysis, DIANA requires to define elasticity parameters. For that, a linear elastic isotropic material model is defined. Essential is the thermal conductivity: 52 W/(m · °C) = 52 T · mm/(s^3 · °C). The thermal capacity is null.

DianaIE

Main menu → Geometry → Assign → Properties [Fig. 7]
Properties → Material → Add material [Fig. 8] → Edit material [Fig. 9]

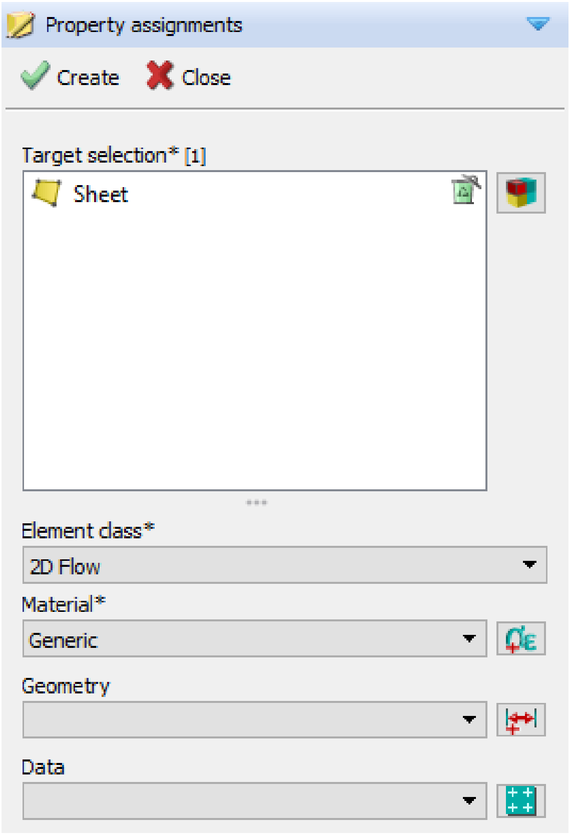

Figure 7: Material property assignment

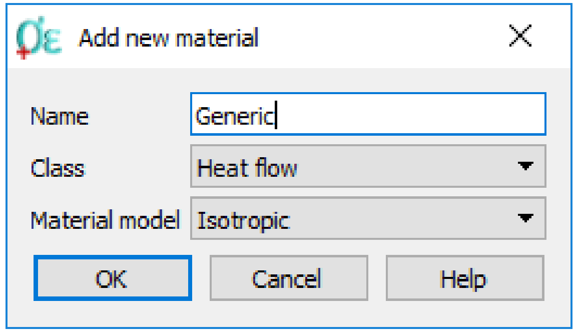

Figure 8: Add new material

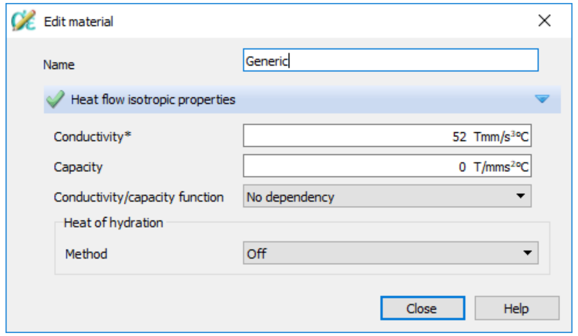

Figure 9: Edit material

2.4 Boundary Conditions

2.4.1 Convection edges

An ambient temperature of 0° is present 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, i.e. a boundary condition, a boundary interface is to be positioned along the aforementioned edges (see Figure 10 and Figure 13) which is treated as a material in order to assign thermal properties to these edges. The interface material is labelled as 'Convection' [Fig. 11].

DianaIE

Main menu → Geometry → Assign → Connection properties [Fig. 10]
Connection properties → Material → Add material [Fig. 11] → Edit material [Fig. 12]
Main menu → Geometry → Assign → Thermal conditions [Fig. 13]

Figure 10: Boundary interface at convection edges

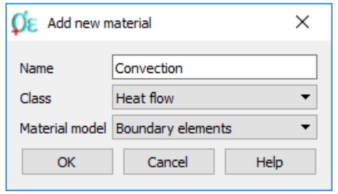

Figure 11: Add new material

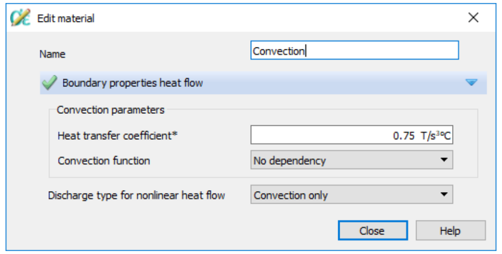

Figure 12: Edit material

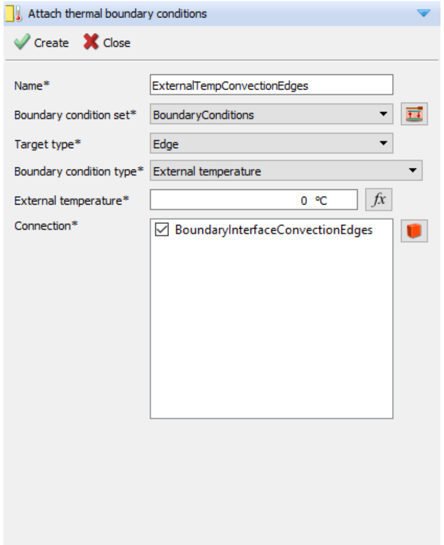

Figure 13: Ambient temperature at convection edges

2.4.2 Bottom edge

At the bottom edge a prescribed temperature of 100° is assigned. Since this temperature is fixed, a two-step procedure needs to be performed in order to specify the boundary condition.

DianaIE

Main menu → Geometry → Assign → Fixed temperatures [Fig. 14]
Main menu → Geometry → Assign → Thermal conditions [Fig. 15]

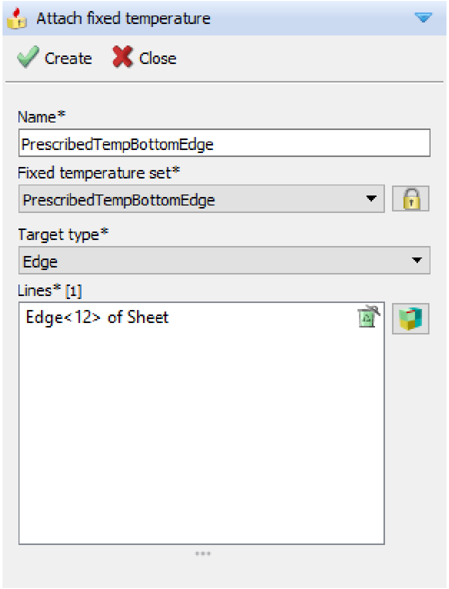

Figure 14: Fixed temperature at bottom edge

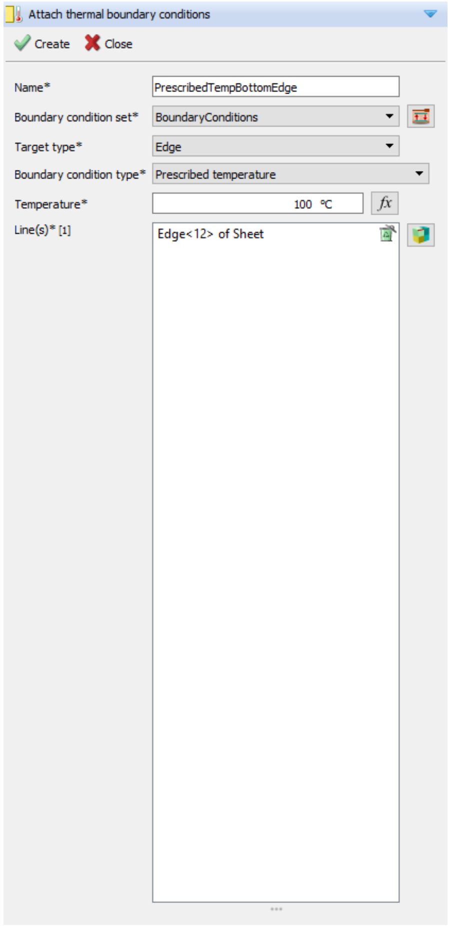

Figure 15: Fixed temperature specified

2.5 Meshing

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. 16] and the left and right edge in five equally sized line segments [Fig. 17]. With this, the geometry is subdivided into fifteen quadrilaterals which is a requirement for this benchmark.

DianaIE

Main menu → Geometry → Assign → Mesh properties [Fig. 16] [Fig. 17]
Main menu → Geometry → Generate mesh

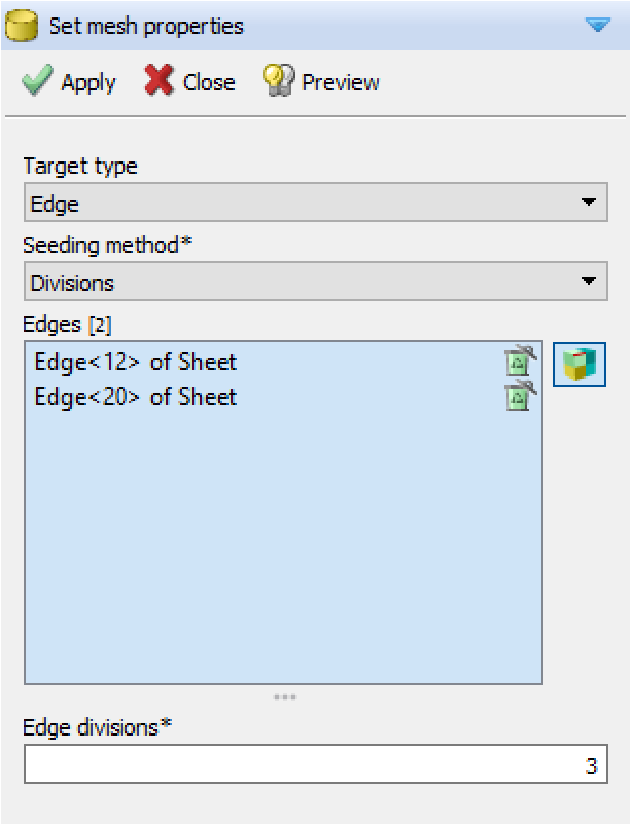

Figure 16: Mesh properties - Dividing the bottom and top edge

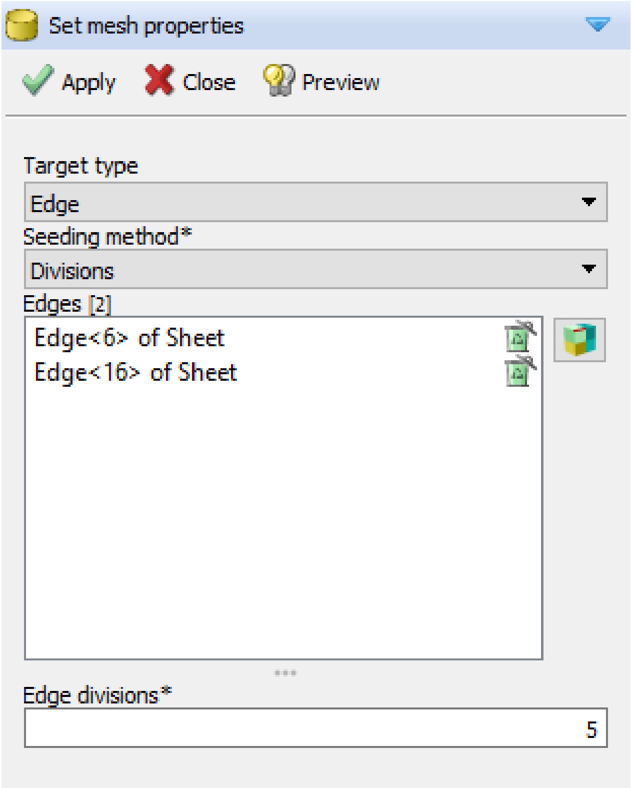

Figure 17: Mesh properties - Dividing the left and right edge

To find out whether or not the target value for the temperature at point E is reached, one needs to know the corresponding node number (in this case it is number 11 [Fig. 18])[2].

DianaIE

Main menu → Mesh → Mesh → Shrunken shading
Main menu → Viewer → Node selection
Graphics window → Select all → Show ids [Fig. 18]

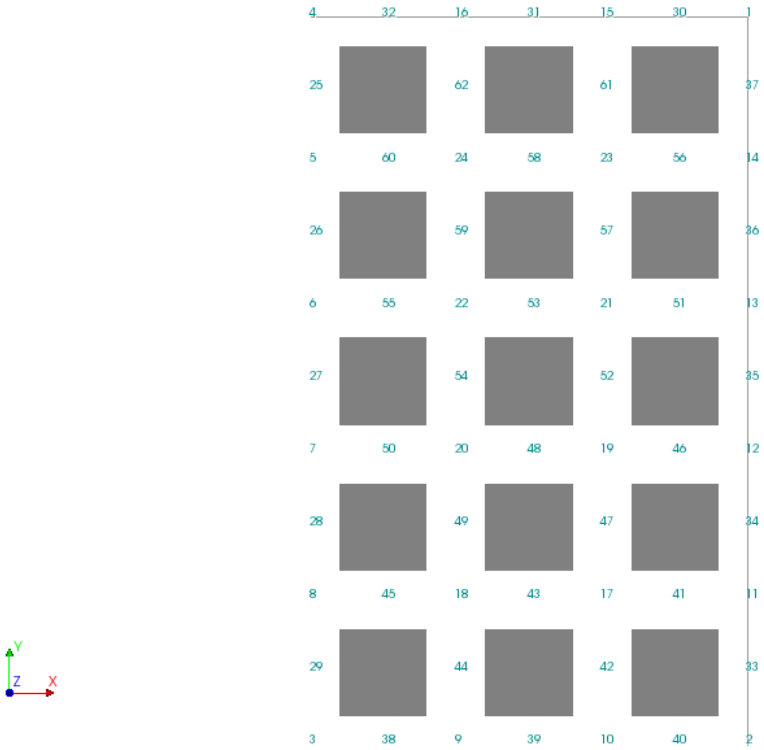

Figure 18: Mesh - Shrunken elements and node identification

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

3 Analysis

3.1 Analysis Commands

We will perform a ‘Steady state heat transfer’ analysis.

DianaIE

Main menu → Analysis → Add analysis
Analysis browser → Analysis1 → Rename → LinSteStaAn [Fig. 19]
Analysis browser → LinSteStaAn → Add command → Steady state heat transfer [Fig. 20] [Fig. 21]
Main menu → Analysis → Run all analyses

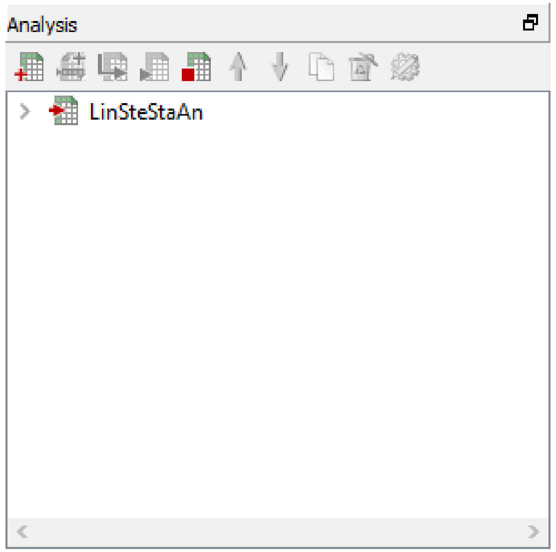

Figure 19: Analysis window

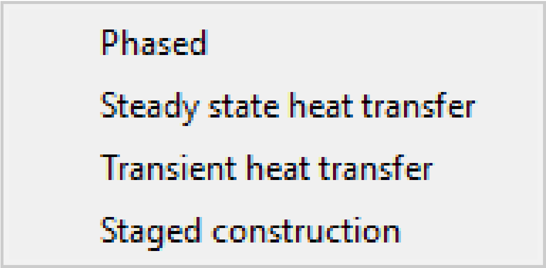

Figure 20: Add command

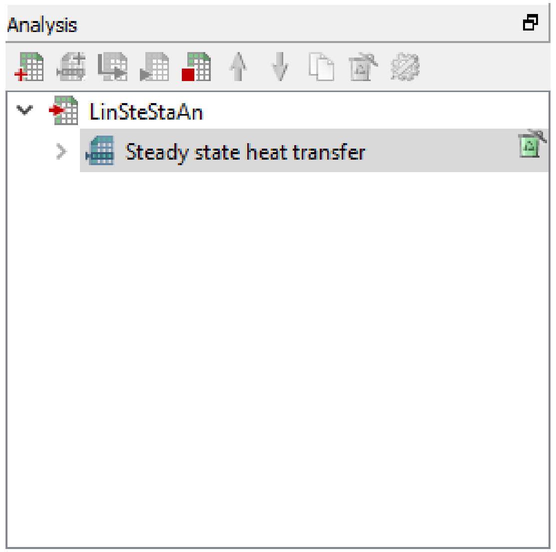

Figure 21: Analysis tree

3.2 Results

3.2.1 Temperature Distribution

The potential temperature distribution PTE is presented [Fig. 23].

DianaIE

Results browser → LinSteStaAn → Output heat transfer analysis → Nodal results → Temperatures → PTE → Show contours [Fig. 22]

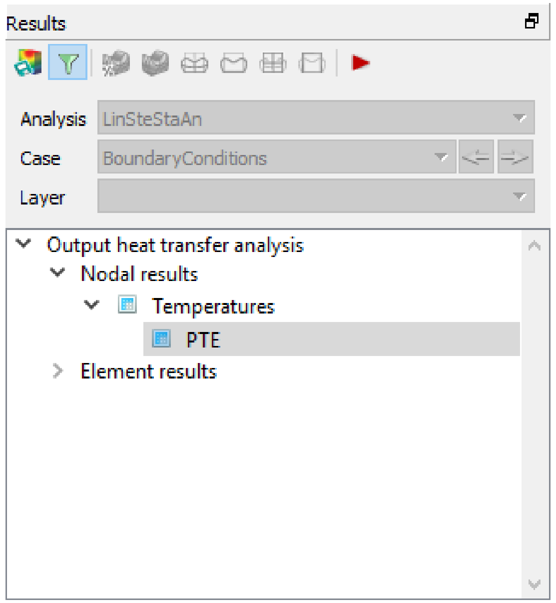

Figure 22: Results browser

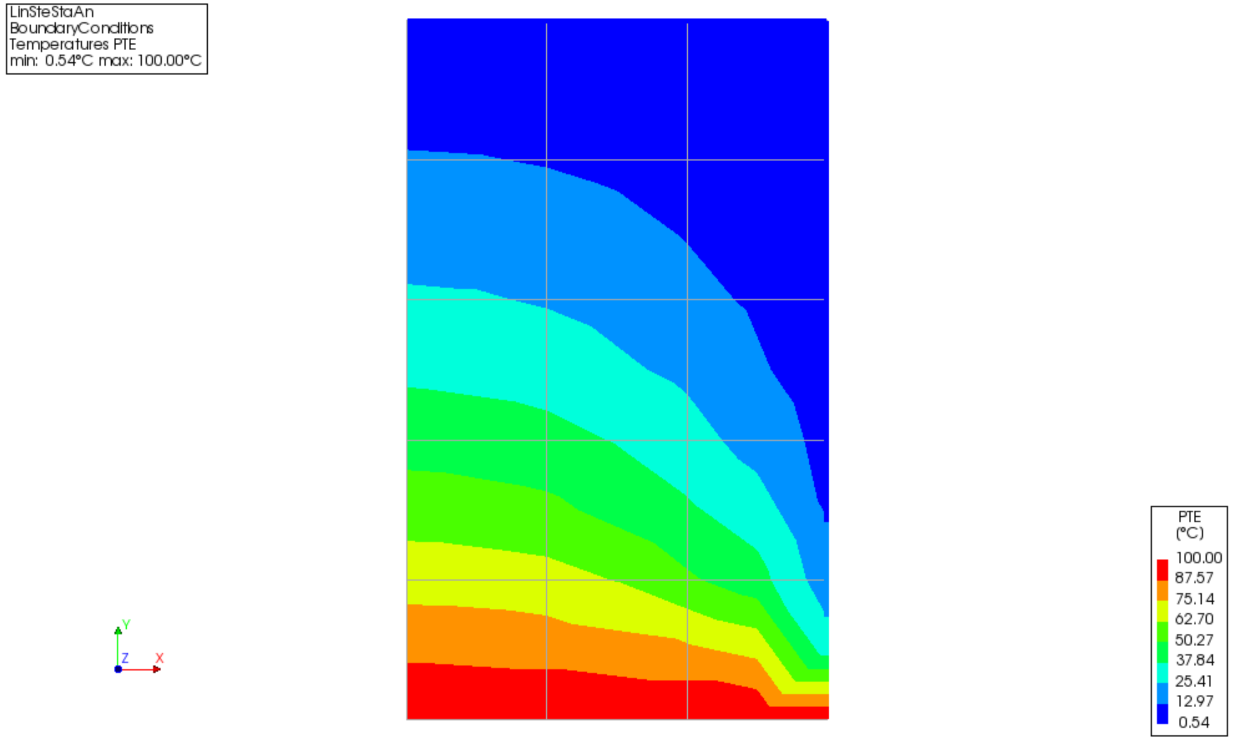

Figure 23: Potential temperature distribution

Results browser → LinSteStaAn → Output heat transfer analysis → Nodal results → Temperatures → PTE → Show table [Fig. 24]

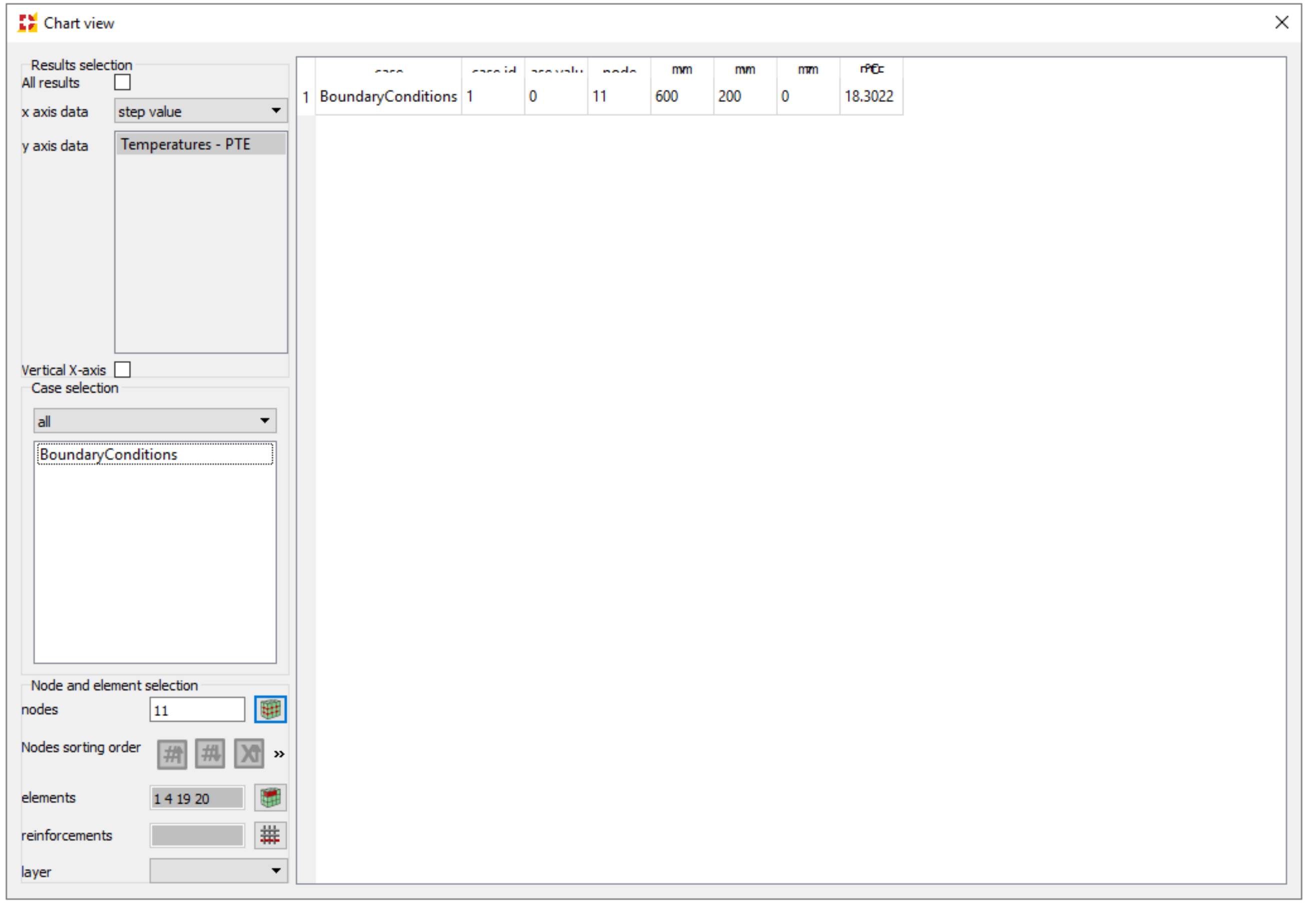

Figure 24: Temperature at point E

Figure 24 shows that the temperature at point E (i.e. node 11) is indeed 18.3 °C; neglecting the 0.0022 °C overestimation.

3.2.2 Heat flux

How the heat flows through the material body can be shown by the heat flux [Fig. 26].

DianaIE

Results browser → LinSteStaAn → Output heat transfer analysis → Element results → Heat Flux → FLXYZ → Show vectors [Fig. 25]

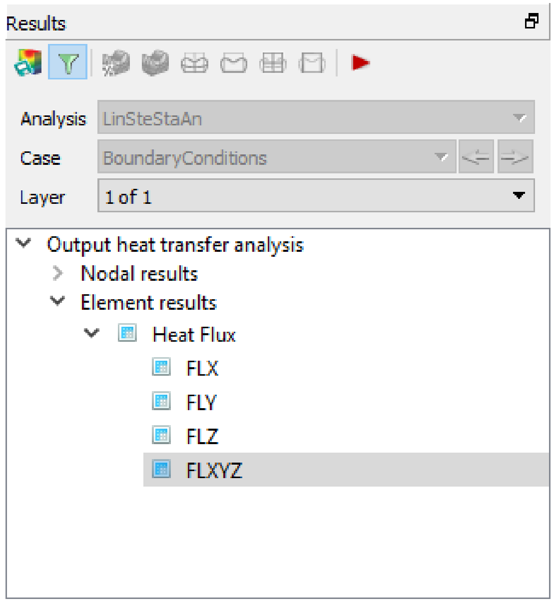

Figure 25: Results browser

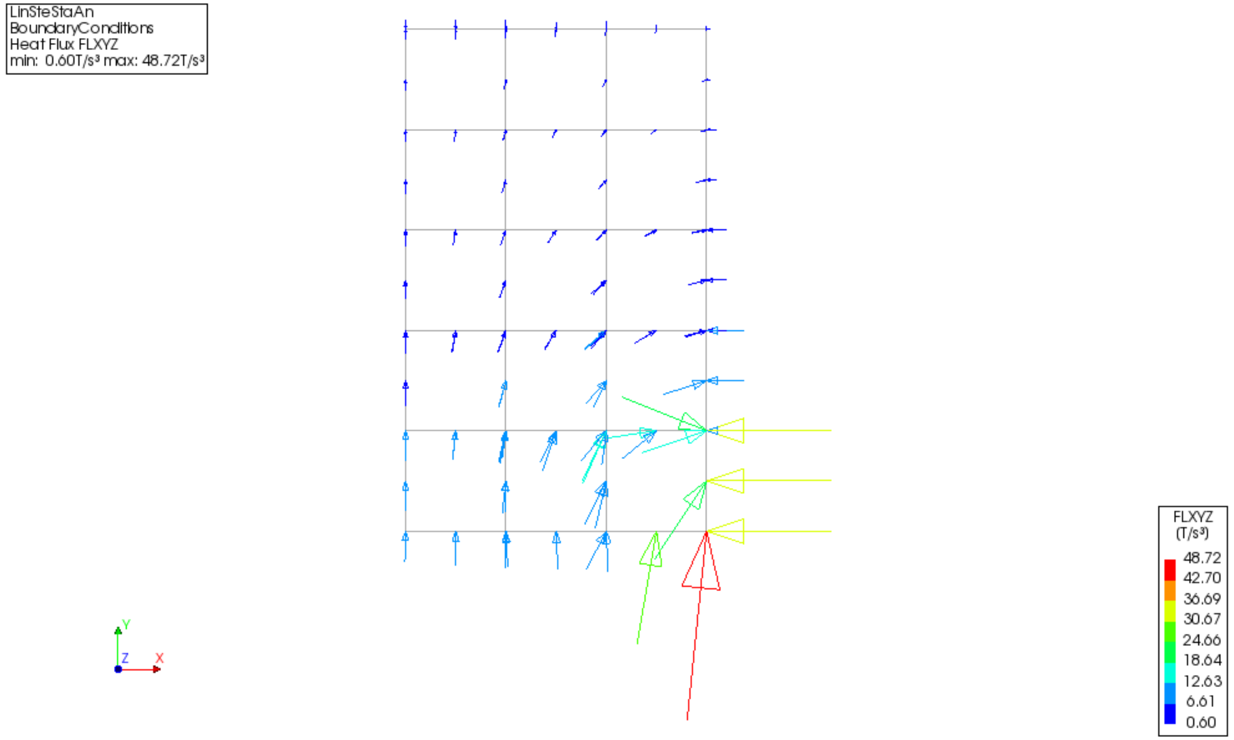

Figure 26: Heat flux through sheet

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.

Appendix A Additional Information

Folder: Tutorials/2DHeatTransfer

Number of elements $\approx$ 15

Keywords:
ANALYS: flow heat steady.
CONSTR: temper.
ELEMEN: bc3ht cq8ht flow potent.
LOAD: elemen node temper.
MATERI: conduc elasti 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.

WWW.DIANAFEA.COM

DIANA FEA BV
Delftechpark 19a
2628 XJ Delft
The Netherlands
T +31 (0) 88 34262 00
F +31 (0) 88 34262 99

DIANA FEA BV
Vlamoven 34
6826 TN Arnhem
The Netherlands
T +31 (0) 88 34262 00
F +31 (0) 88 34262 99