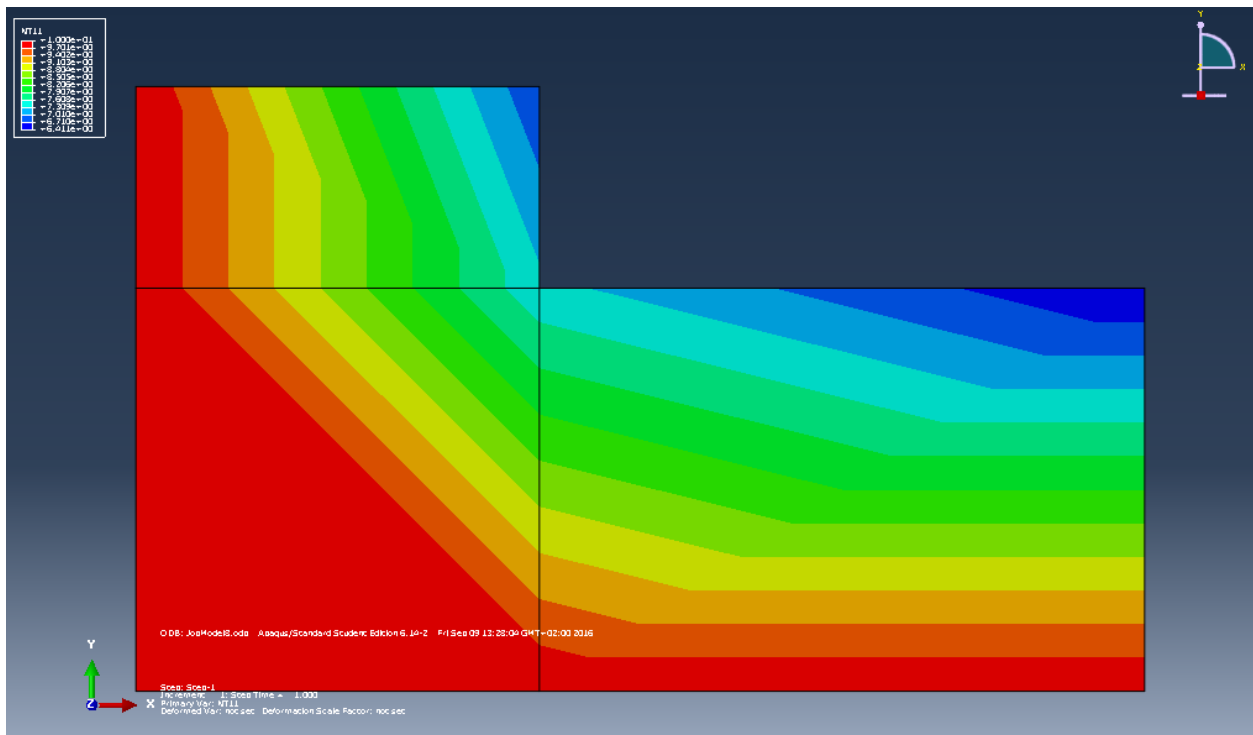




# “Heat transfer analysis in ABAQUS”

Marzena Mucha



Free student version of ABAQUS is available on website: <https://academy.3ds.com/en/software/abaqus-student-edition>

There are no imposed units in ABAQUS, so be very careful with input values.

## Problem formulation

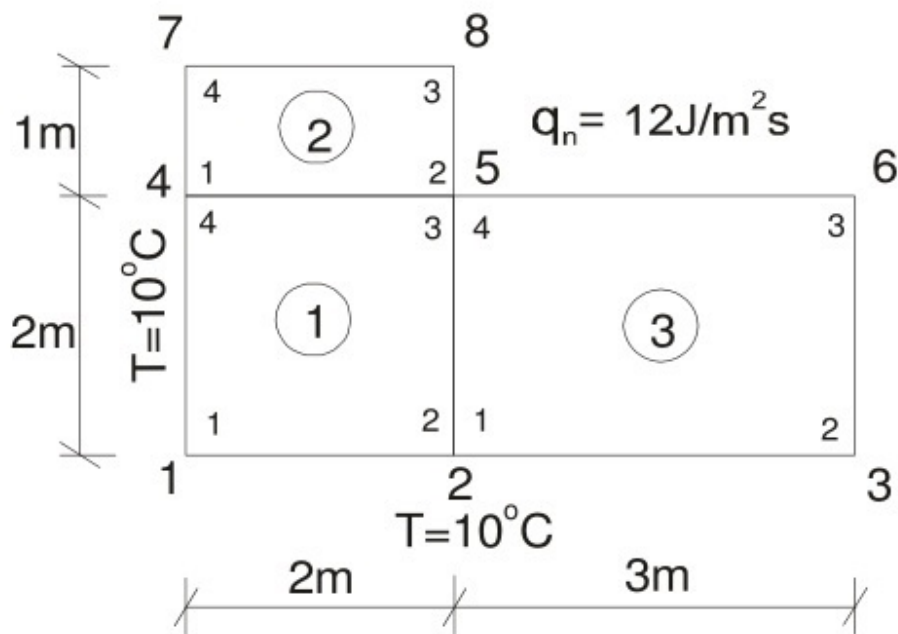
Input:

Material type: orthotropic

Heat flow factors:  $k_x = 4 \frac{J}{\text{°C}\cdot\text{m}\cdot\text{s}}$ ,  $k_y = 7 \frac{J}{\text{°C}\cdot\text{m}\cdot\text{s}}$

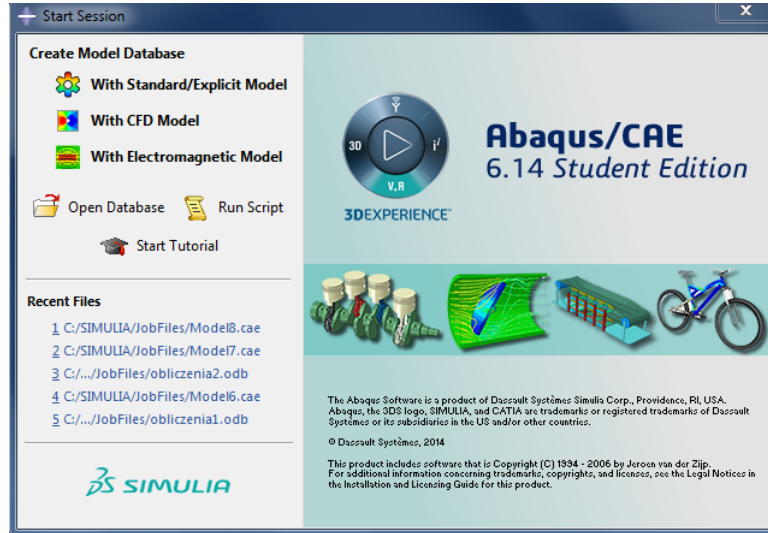
Dirichlet boundary conditions (temperature): edge 1-7 and 1-3, value  $T = 10^\circ\text{C}$

Neumann boundary conditions: edge 5-8 and 5-6, value  $q_n = 12 \frac{J}{\text{m}^2\cdot\text{s}}$

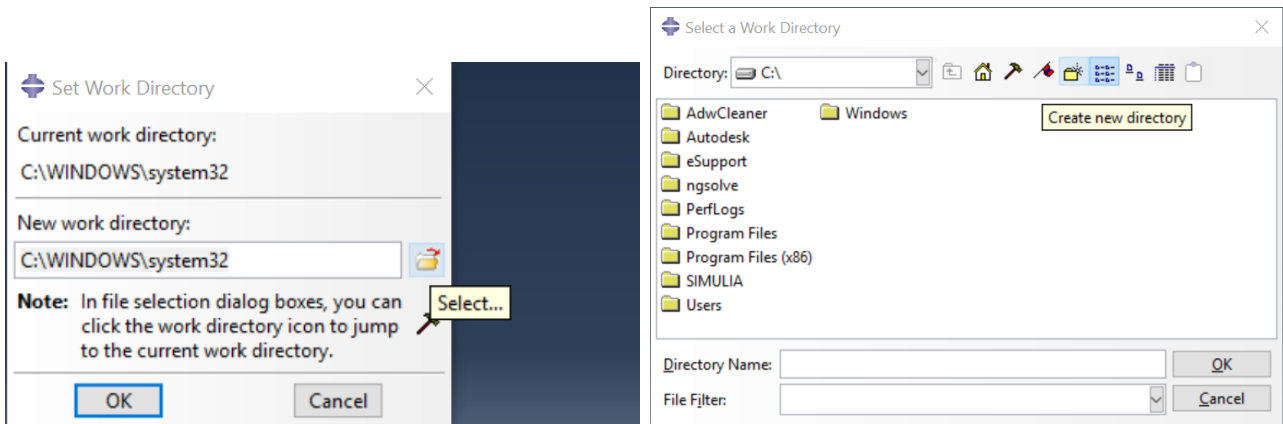


# Solution

Run ABAQUS (Abaqus 6.11 Student Edition) and choose Create Model Database With Standard/Explicit Model



Choose File and Set Work Directory... . In window Set Work Directory choose Select... on the right hand side and create new folder on your disk (icon Create new directory). Choose this folder and click button OK. In window Set Work Directory you should see new path to your folder, click button OK.

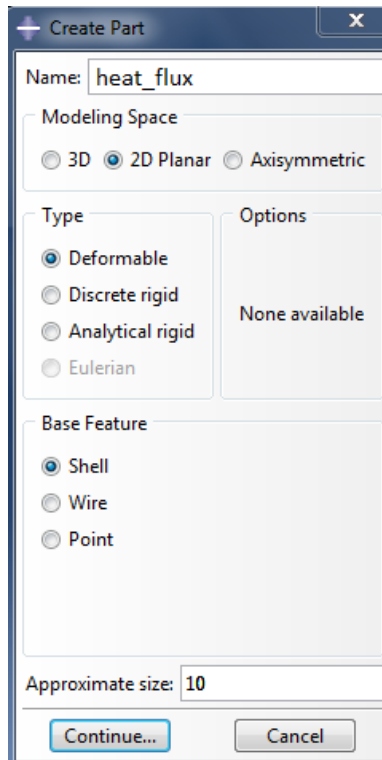


## 1 Part

On the left-hand side bar choose icon Create Part.



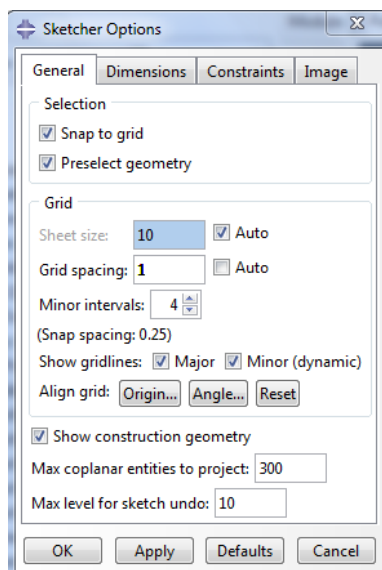
In window Create Part write Name: heat\_flux, choose 2D Planar and Approximate size 10. Click button Continue...



On the left-hand side choose icon Sketcher Options:



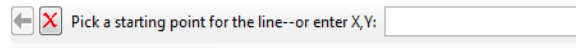
In window Sketcher Options change Grid spacing to 1 and click OK.



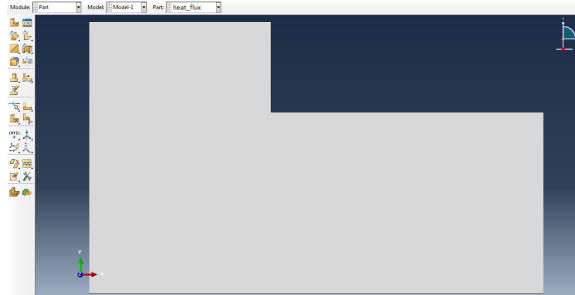
Next choose icon Create Lines: Connected.



At the bottom, under working area enter point coordinates: 0,0 Enter 0,3 Enter 2,3 Enter 2,2 Enter 5,2 Enter 5,0 Enter 0,0 Enter. End by clicking in the sign **X** and confirm by button Done.



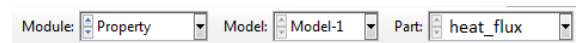
Part heat\_flux was created.



**Save your project!** To JobMesh and report\_mesh names add \_Surname for example: JobMesh1\_Kowalski, report\_mesh1\_Kowalski. Avoid special signs like polish letters.

## 2 Property

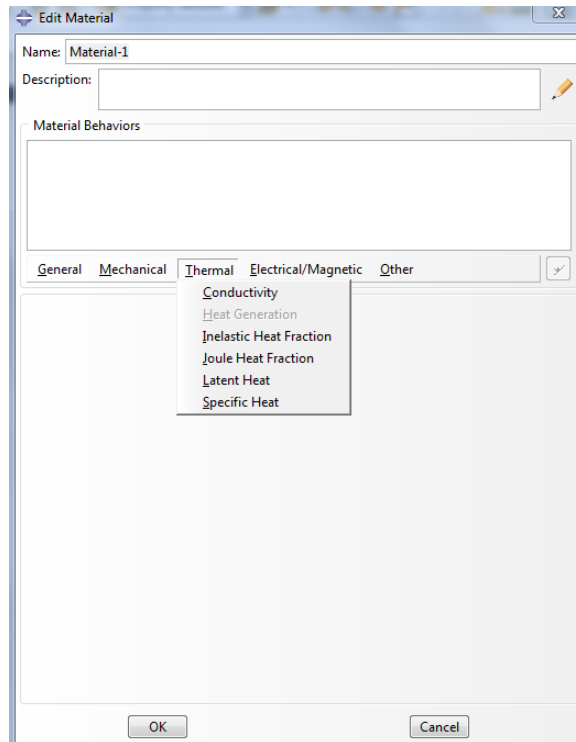
Change Module to Property.



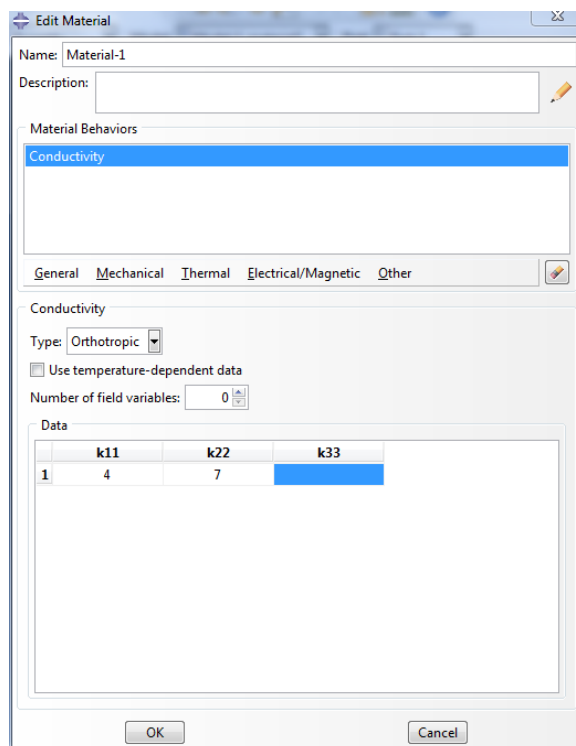
Choose icon Create Material.



In window Edit Material create Material-1. In list Thermal choose Conductivity.



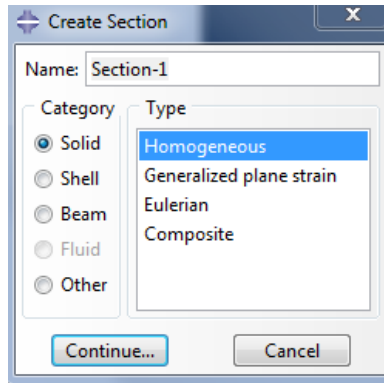
Next change Type to orthotropic and enter  $k_x$  and  $k_y$  as is shown in the picture. Click button OK.



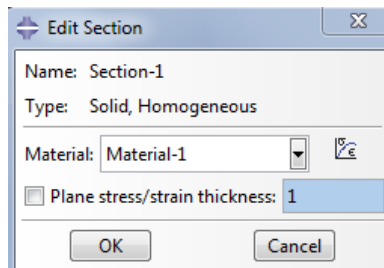
Choose icon Create Section.



In window Create Section choose Solid and Homogenous and press button Continue...



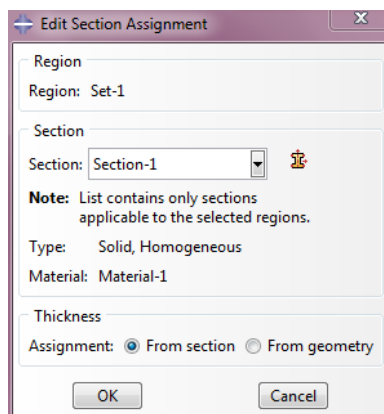
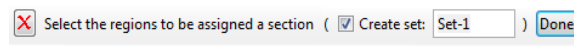
In the next window confirm by pressing button OK.



Choose icon Assign Section.



Select part heat\_flux and confirm by pressing button Done. Next in window Edit Section Assignment press button OK. Material-1 has been assigned to part heat\_flux.

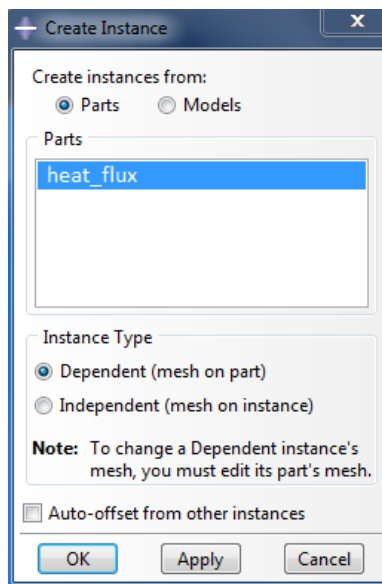


### 3 Assembly

Change Module to Assembly and press icon Create Instance.



In window Create Instance choose Dependent (mesh on part) and press button OK.



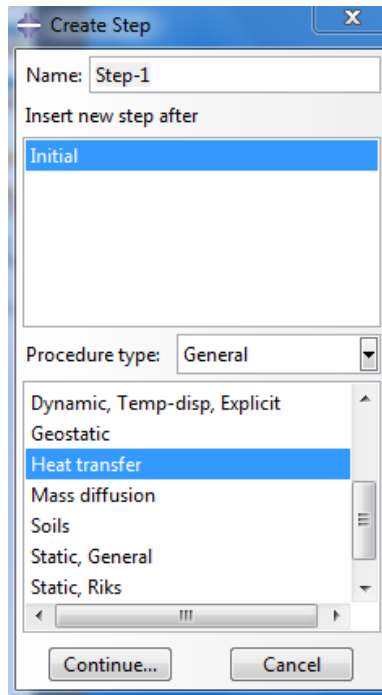
### 4 Step

Change Module to Step and click icon Create Step.

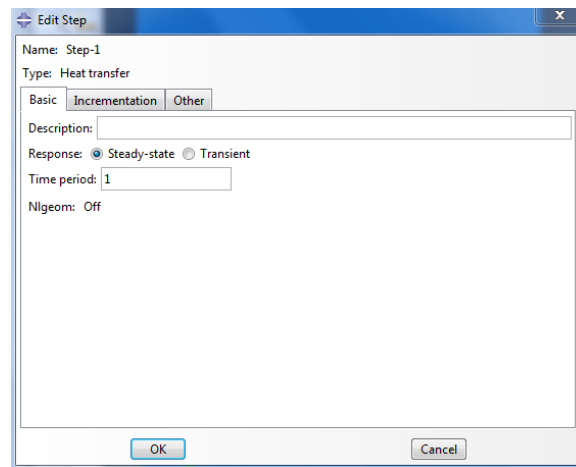


In window Create Step create Step-1, choose Heat transfer and press button Continue...

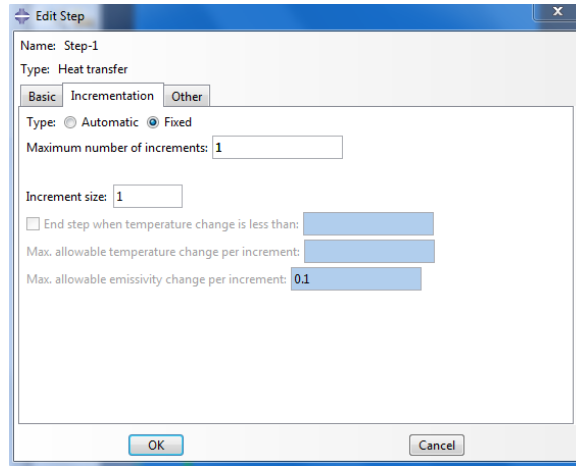




Next in Edit Step choose tab Basic and mark Steady-State. If warning (Default load variation with time has been changed to Ramp linearly over step) appear click Dismiss.



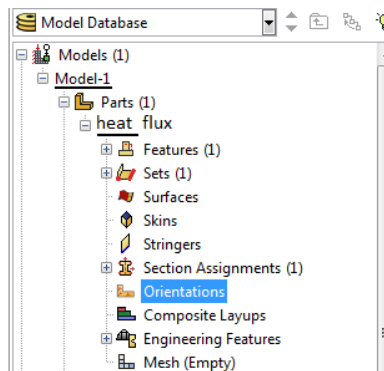
In tab Incrementation mark Type Fixed and change value Maximum number of increments to 1.



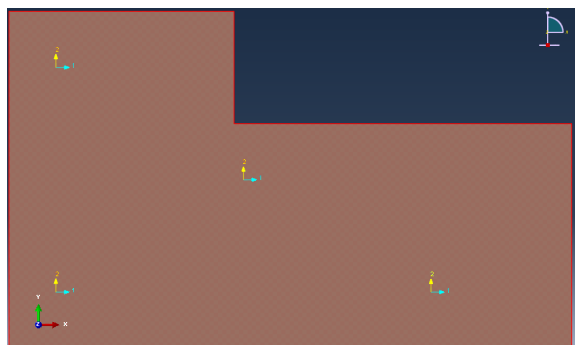
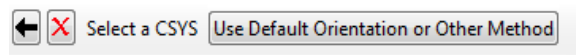
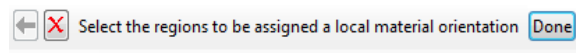
Press button OK.

## 5 Orientations

Expand sidelist as is shown in the picture below and choose Orientations (two clicks).

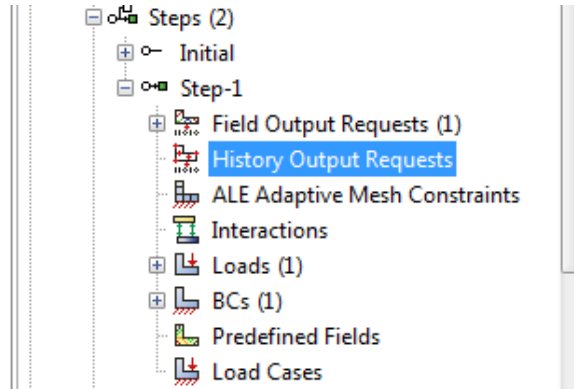


Mark heat\_flux area, press button Done, next Use Default Orientations or Other Method and then in window Edit Material Orientation press button OK.

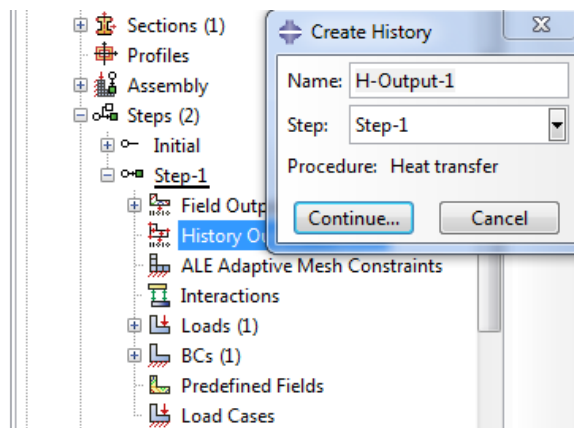


## 6 History Output Request

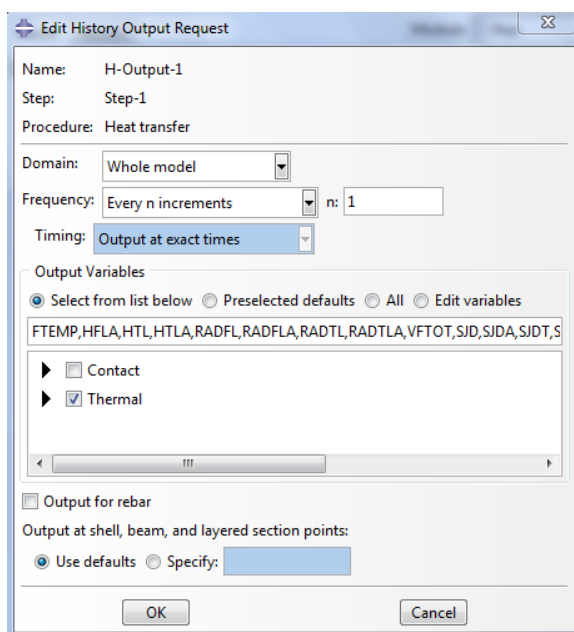
Expand sidelist as is shown in the picture below and choose History Output Request (two clicks).



In window Create History press button Continue..

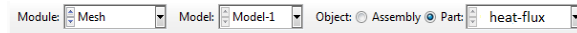


In window Edit History Output Request mark Thermal and press button OK.

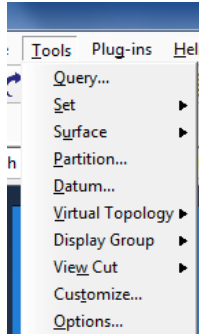


## 7 Mesh

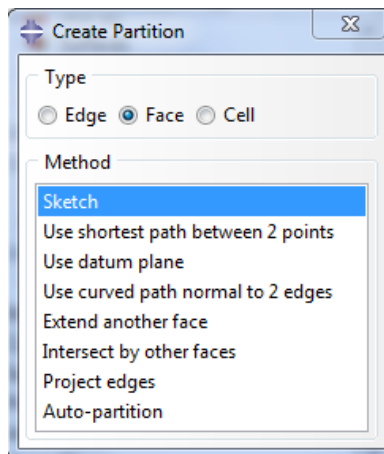
Change Module to Mesh and **Object to Part**.



In Menu bar choose Tools → Partition...



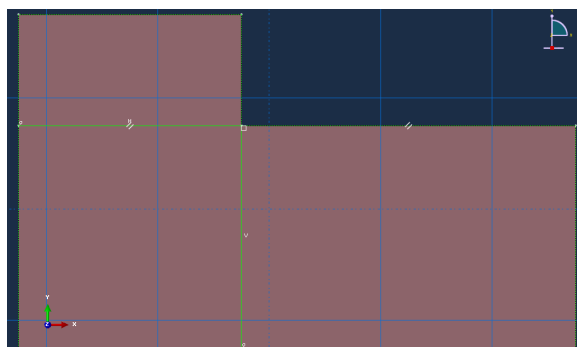
In window Create Partition choose Type Face and Method Sketch.



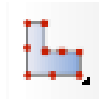
Choose icon Create Lines: Connected.



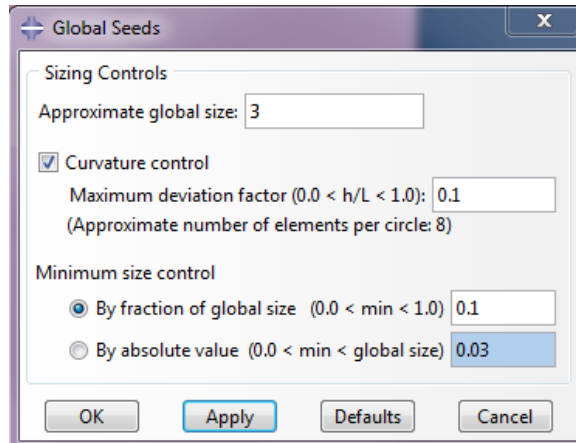
and divide part heat\_flux into regions as is shown in the picture below. **First enter horizontal line, then vertical line.** To end one segment press Esc button. To end press button Done and then **X** at the bottom of the screen. You can close window Create Partition.



Choose icon Seed Part.



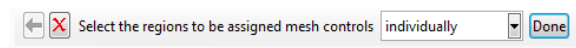
In window Global Seeds set Approximate global size to 3 and press button OK.



Choose icon Assign Mesh Controls.

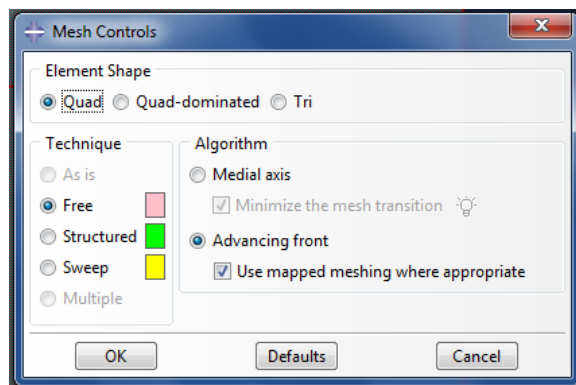


Mark all regions and press button Done.



In window Mesh Controls choose Element Shape Quad and Technique Structured. Press button OK.

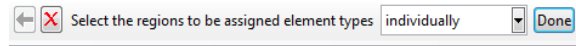
**For triangle elements choose Tri instead of Quad.**



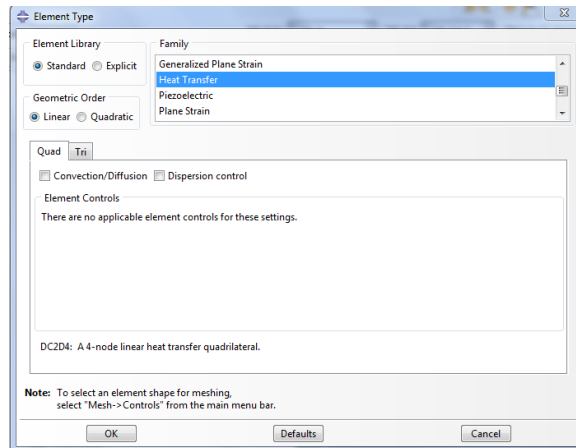
Choose icon Assign Element Type.



Mark all regions and press button Done.



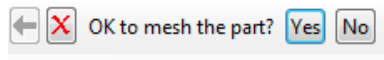
In window Element Type choose Family Heat Transfer and press button OK.



Choose icon Mesh Part t (or in Menu Mesh → Part....)

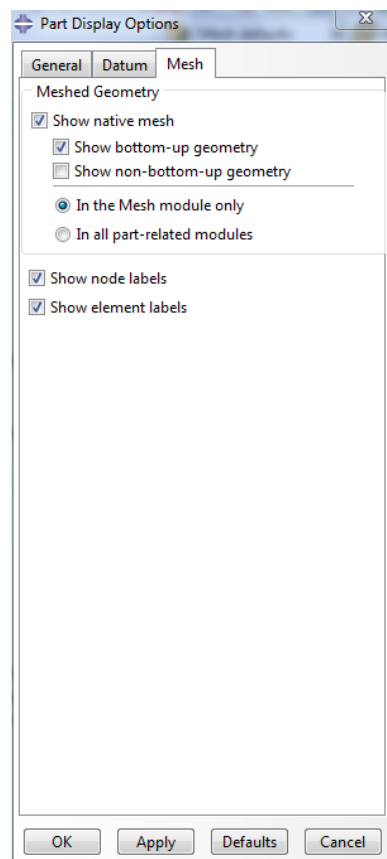
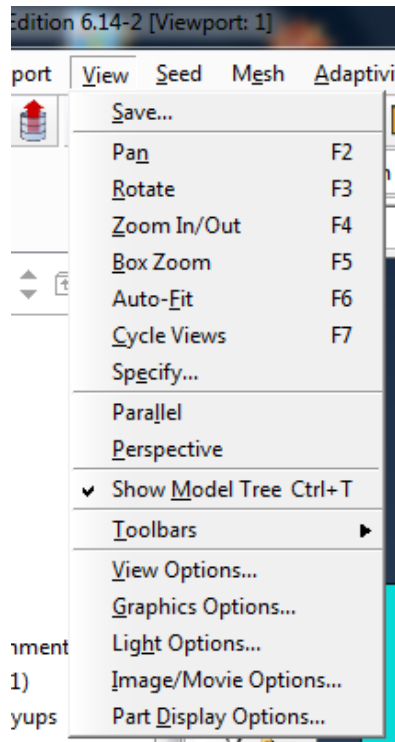


Press button Yes.

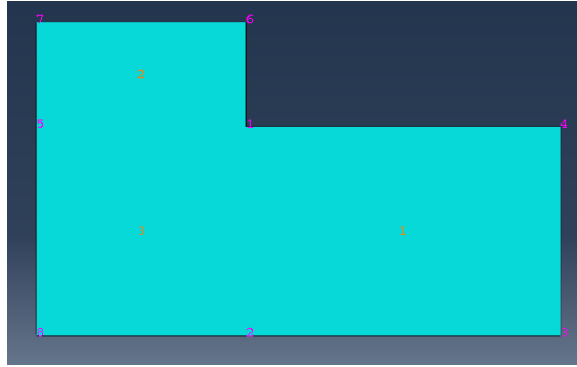


Three elements have been created. Message: "3 elements have been generated on part: heat\_flux" is on the bottom of the screen and color of the region is changed to blue.

To see numbering of elements and nodes from Menu bar View choose Part Display Options (at the bottom of the list). In tab Mesh mark Show node labels and Show element labels. Press button OK.

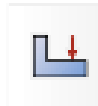


You should now see numbers on the screen.

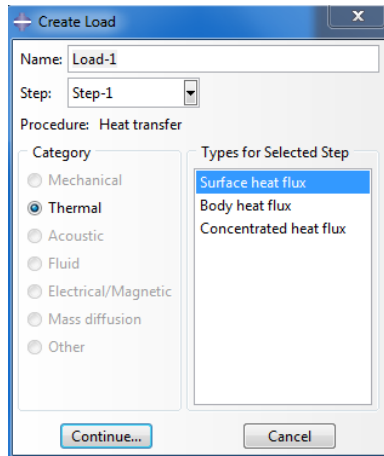


## 8 Load

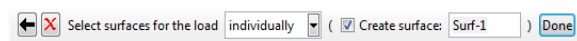
Change Module to Load and press icon Create Load.



In window Create Load choose Surface heat flux (Neumann boundary conditions), next press button Continue...

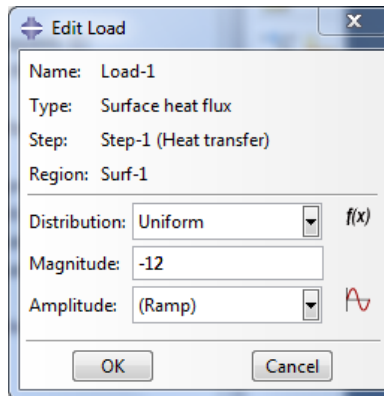


Mark edges 5-8 and 5-6, (pressing button Shift you can mark more than one edge at the same time), on which boundary conditions should be specified and press button Done at the bottom of the screen.

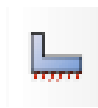


In window Edit Load in field Magnitude write -12. Click button OK.

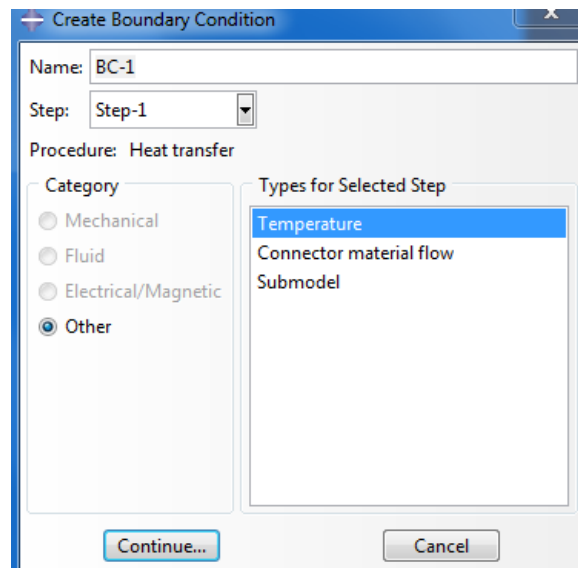




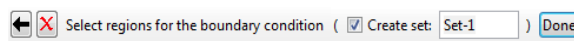
Choose icon Create Boundary Condition.



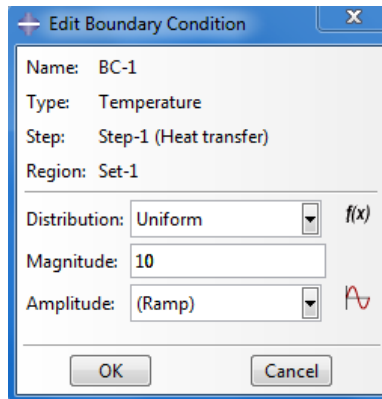
In window Create Boundary Conditions choose Type for Selected Step: Temperature (Dirichlet boundary conditions), next press button Continue...



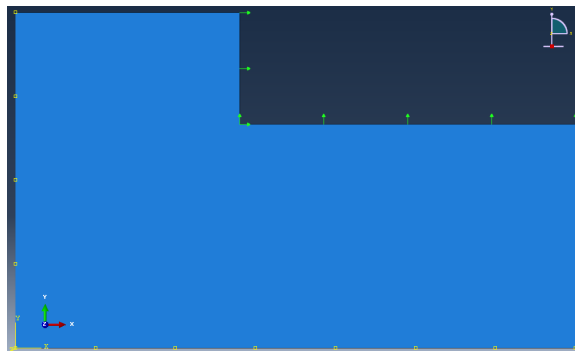
Mark edges (edges 1-7 and 1-3), on which boundary conditions should be defined and press button Done at the bottom of the screen.



In window Edit Boundary Conditions in field Magnitude write 10. Press button OK.



Boundary conditions have been added to part heat\_flux.

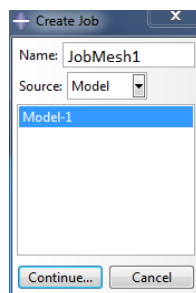


## 9 Job

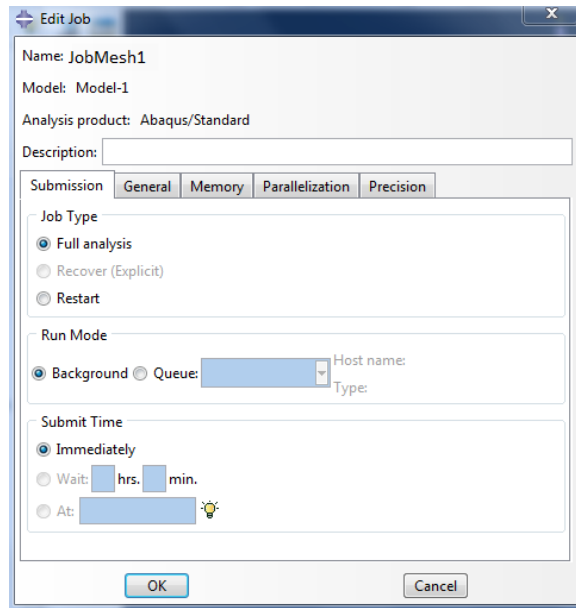
Change Module to Job and choose icon Create Job.



In window Create Job write JobMesh1 and press button Continue...



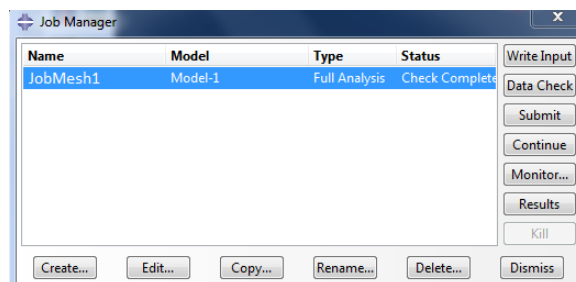
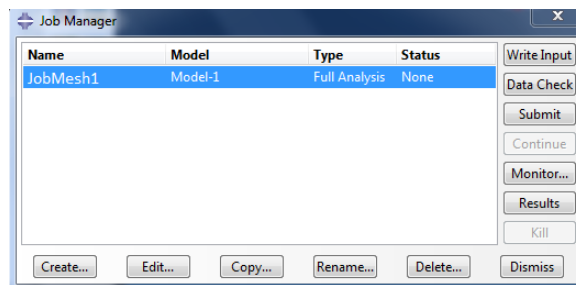
In window Edit Job press button OK.



Choose icon Job Manager.



In window Job Manager press button Data Check (if warning appears press button OK). Now Abaqus will check your model, if there are no mistakes you can press button Submit (if warning appears press button OK).



If the program finds no problems the status should change to Completed.

## 10 Visualization

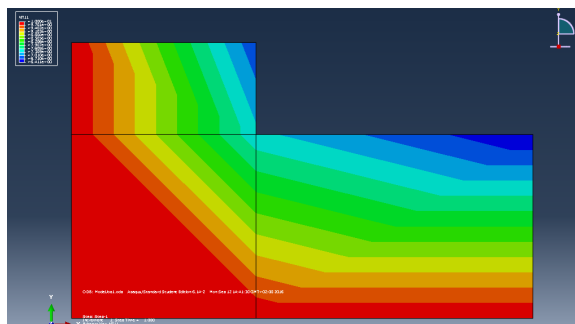
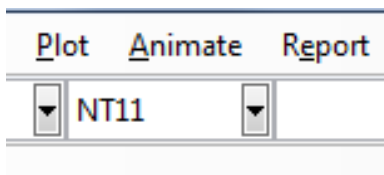
In window Job Manager press button Results, you will be automatically moved to Visualization module.



Choose icon Plot Contours on Deformed Shape.

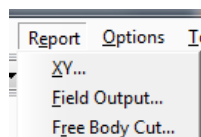


From the list choose NT11 (temperature).

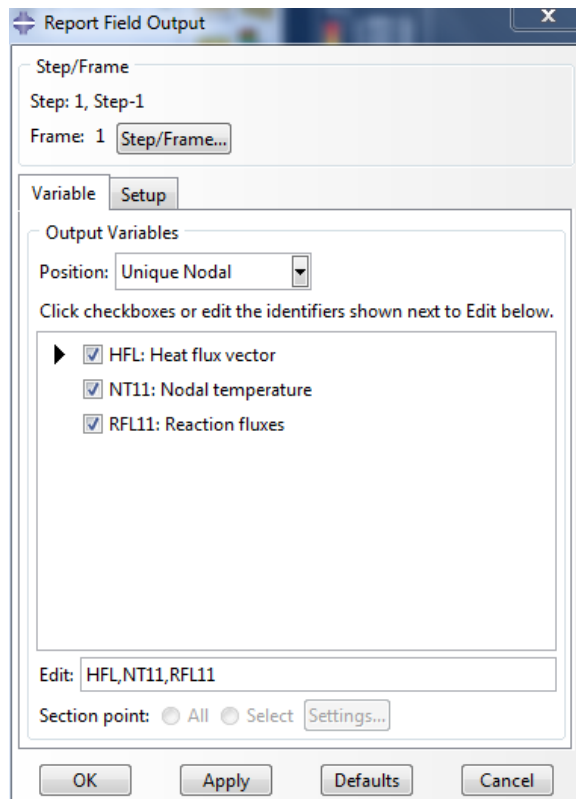
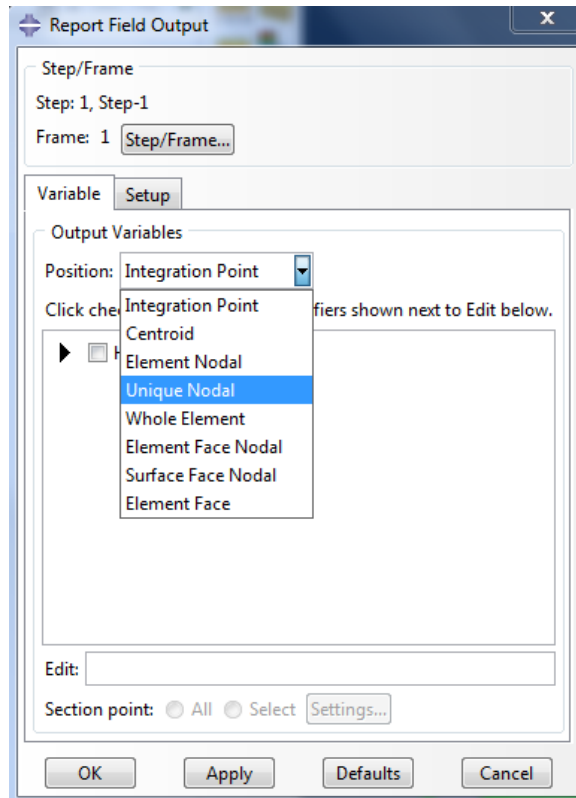


## 11 Report

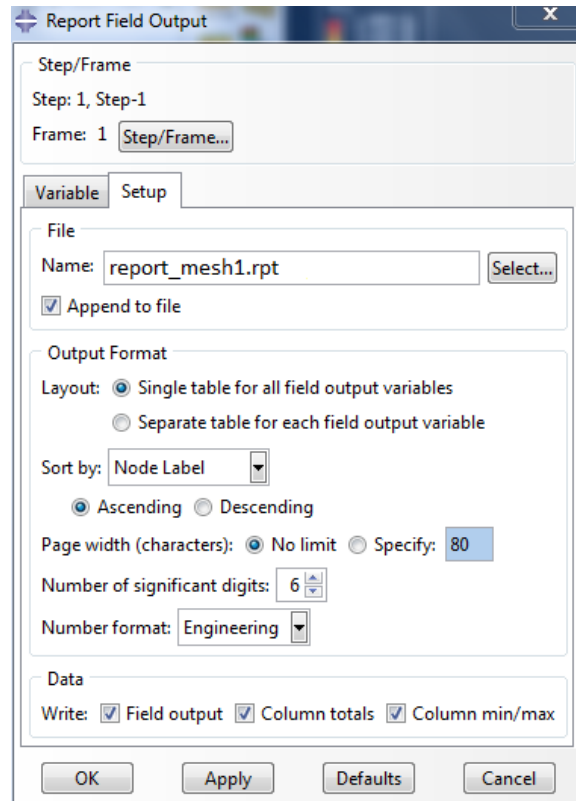
In menu Report choose Field Output...



In window Report Field Output, tab Variable in list Position set Unique Nodal and tick check-boxes HFL (heat flux vector), NT11 (Nodal temperature) and RFL11 (Reaction fluxes).



In tab Setup set file name (Name) as report\_mesh1. Click Select... if you want to choose your own folder.



Click button OK.

Report can be viewed in Windows Notepad (Notatnik).

```

report_mesh1.rpt - Notatnik
Plik  Edycja  Format  Widok  Pomoc
*****
Field output Report, written Thu Sep 15 00:00:20 2016
Source 1
-----
      ODB: C:/SIMULIA/JobFiles/JobMesh1.odb
      Step: Step-1
      Frame: Increment      1: Step Time =      1.000

Loc 1 : Nodal values from source 1
Output sorted by column "Node Label".

Field output reported at nodes for part:      HEAT FLUX-1
Computation algorithm: EXTRAPOLATE_COMPUTE_AVERAGE
Averaged at nodes
Averaging regions: ODB_REGIONS

      Node      NT11      RFL11      HFL.Magnitude      HFL.HFL1      HFL.HFL2
      Label      @Loc 1      @Loc 1      @Loc 1      @Loc 1      @Loc 1
-----
      1          7.38704          0.          8.24836          3.54728          6.86191
      2          10.          20.2139          8.58379          689.634E-03          8.53984
      3          10.          16.6991          11.8417          274.902E-03          11.8385
      4          6.41140          0.          11.8828          1.02595          11.8385
      5          10.          4.74197          5.06312          4.80794          1.43604
      6          6.75197          0.          7.14674          6.22765          3.50605
      7          10.          1.55451          6.29810          6.22765          939.443E-03
      8          10.          4.79042          2.22592          1.10437          1.93264

      Minimum      6.41140          0.          2.22592          274.902E-03          939.443E-03
      At Node      4          6          8          3          7
      Maximum      10.          20.2139          11.8828          6.22765          11.8385
      At Node      8          2          4          7          4
      Total      70.5504          48.0000          61.2905          23.9054          46.8929
    
```

## 12 Temperature at any point

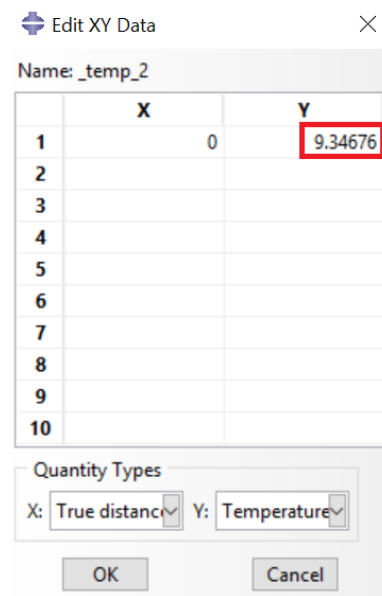
In case of check temperature in any point from menu choose Tools → Path → Create. Next in window Create Path choose Point list and create Path\_mesh(number). In window Edit Point

List Path write coordinates of the point for example 1,1,0 and confirm by pressing button Ok. Next choose from menu Tools → XY Date → Create. In window Create XY mark Path and click button Continue... In window XY Data from Path click Plot. To go back to former view click icon:



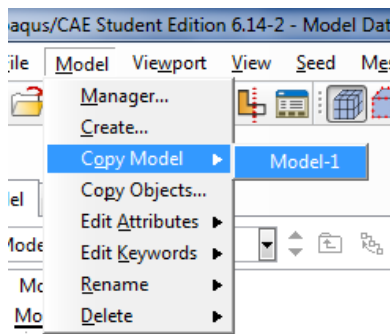
To check exact solution choose Tools → XY Data → Edit.

Results for first mesh:

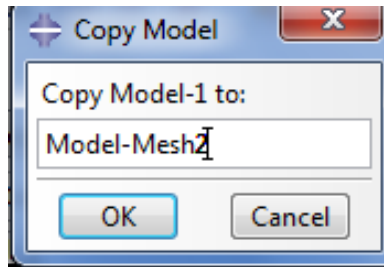


## 13 Creating finer mesh, Mesh2

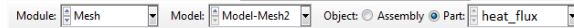
Change Module to Mesh. In menu Model choose Copy Model, Model-1.



Create Model-Mesh2 by clicking OK.



Check if Model-Mesh2 is reported as active.



Remove mesh. In menu Mesh choose Delete Part Native Mesh... (Delete Part Mesh) and press button Yes at the bottom of the screen.

Press icon Seed Part. In window Global Seeds change Approximate global size to 0.5.

In menu Mesh choose Part... and press button Yes at the bottom of the screen.

Change Module to Job and choose icon Create Job.

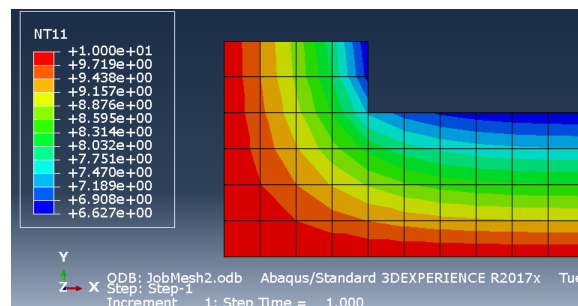
In window Create Job write JobMesh2 (mark Model-Mesh2) and press button Continue... and in window Edit Job press button OK.

Choose icon Job Manager.

In window Job Manager press button Data Check. Now program will check your model, if there are no mistakes you can press button Submit.

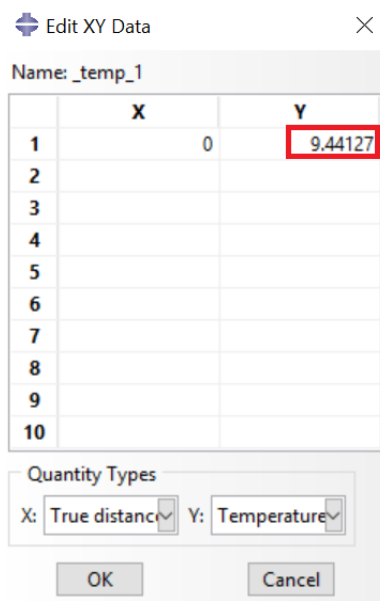
If the program finds no problems the status should change to Completed. The rest of the steps are repeated as for first mesh.

Create new report, report\_\_mesh2.



Remember about checking temperature in the same point as for mesh1. Results for second mesh:





## 14 Creating finer mesh, Mesh3

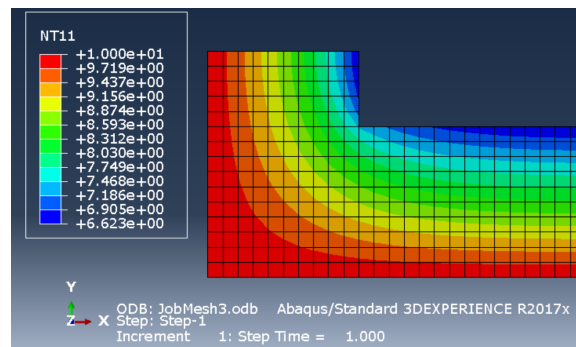
Repeat steps from point 12.

Create Model-Mesh3 (copy Model-Mesh1).

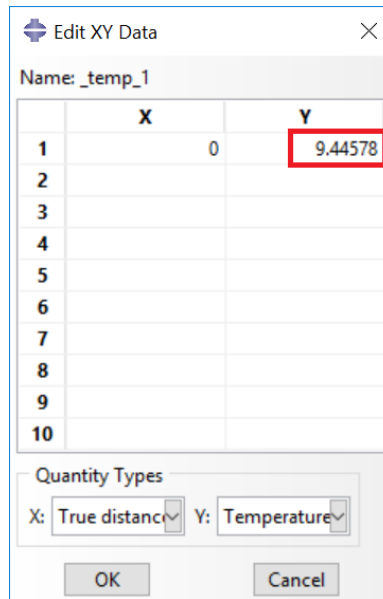
In window Global Seeds change Approximate global size to 0.2.

In window Edit Job write JobMesh3 and perform computations.

Create new report, report\_mesh3.



Remember about checking temperature in the same point as for mesh1. Results for third mesh:



## 15 Project loading

In the case of loading the project during the next class choose tab Visualization and in menu File choose Open. You should have three files: JobMesh1, JobMesh2 and JobMesh3. Choosing one of them will load the results for the respective case (without a model). To load the model go back to tab Part and perform the upload.