Obliczenia inżynierskie w chmurze

Ryszard Zawiła

299172

Final report

FEA analysis of Beam structure using AZURE COULD COMPUTING and PYTHON

Warszawa, 2022

1. Introduction

The purpose of this project is to carry out FEA analysis using AZURE cloud computing. The second objective is to familiarize with the procedure of setting up the virtual machine and using it to perform calculations that a given person needs.

Nowadays during the design process of many machines and devices engineers will be performing a variety of different simulations. Sometimes the available machines in the company are not able to complete calculations in reasonable time, so then a cloud computing becomes an option. It gives an opportunity for the engineers to move calculations to the big machines which services can be "rented". This can greatly reduce the time needed for big analysis thus speeding the entire design process

2. Virtual Machine set-up

Machine was set up according to provided instruction. Following picture summarizes the machine parameters.

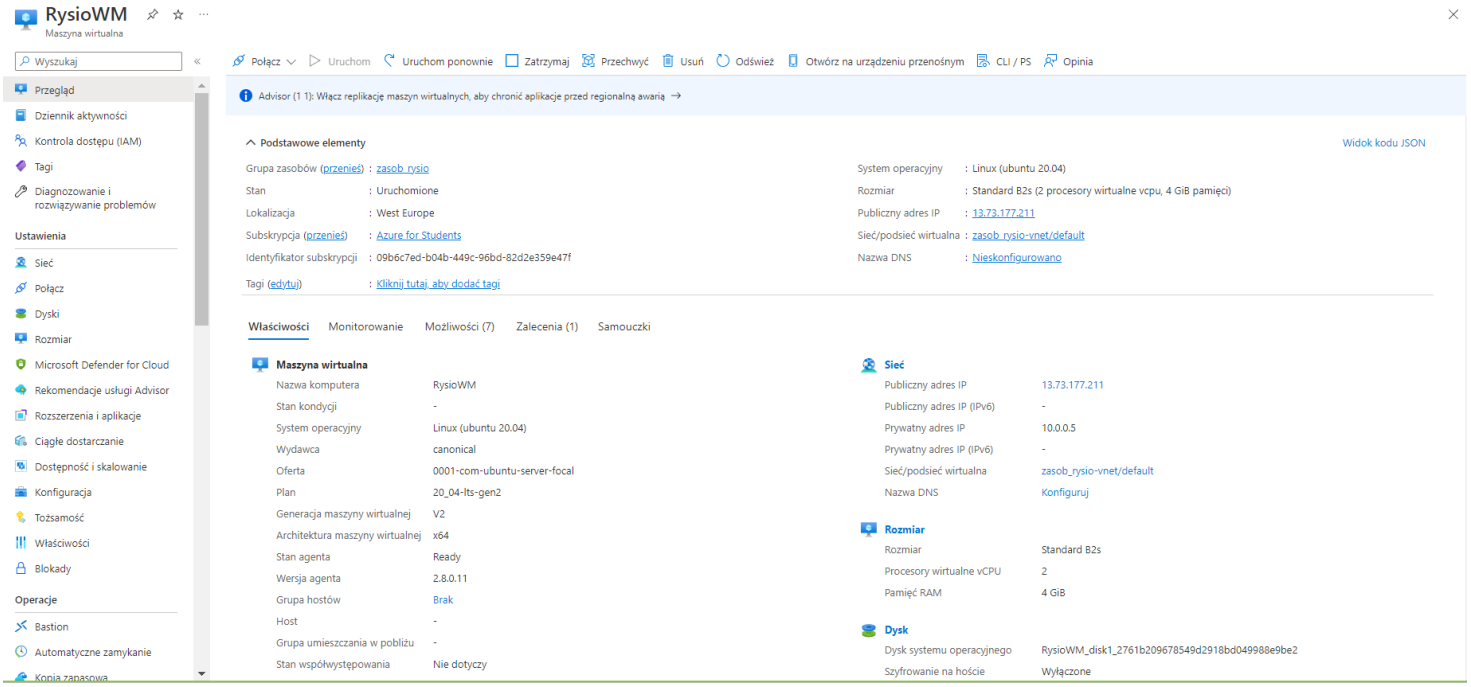

Figure 2-1 - Virtual Machine set-up

Ubuntu 20.04 was chosen for this machine. The access will be done by port 22 with the use of SSH key and such connection was established. Putty interface was used for establishing the connection and executing the commands on the virtual machine.

3. Analysed model

To perform the calculation using cloud computing a python code was prepared. It used solidspy library. It contains necessary commands for the 2D FEA analysis. The following picture shows the meshed model that was analysed. The mesh was done by the additional software called GMSH.

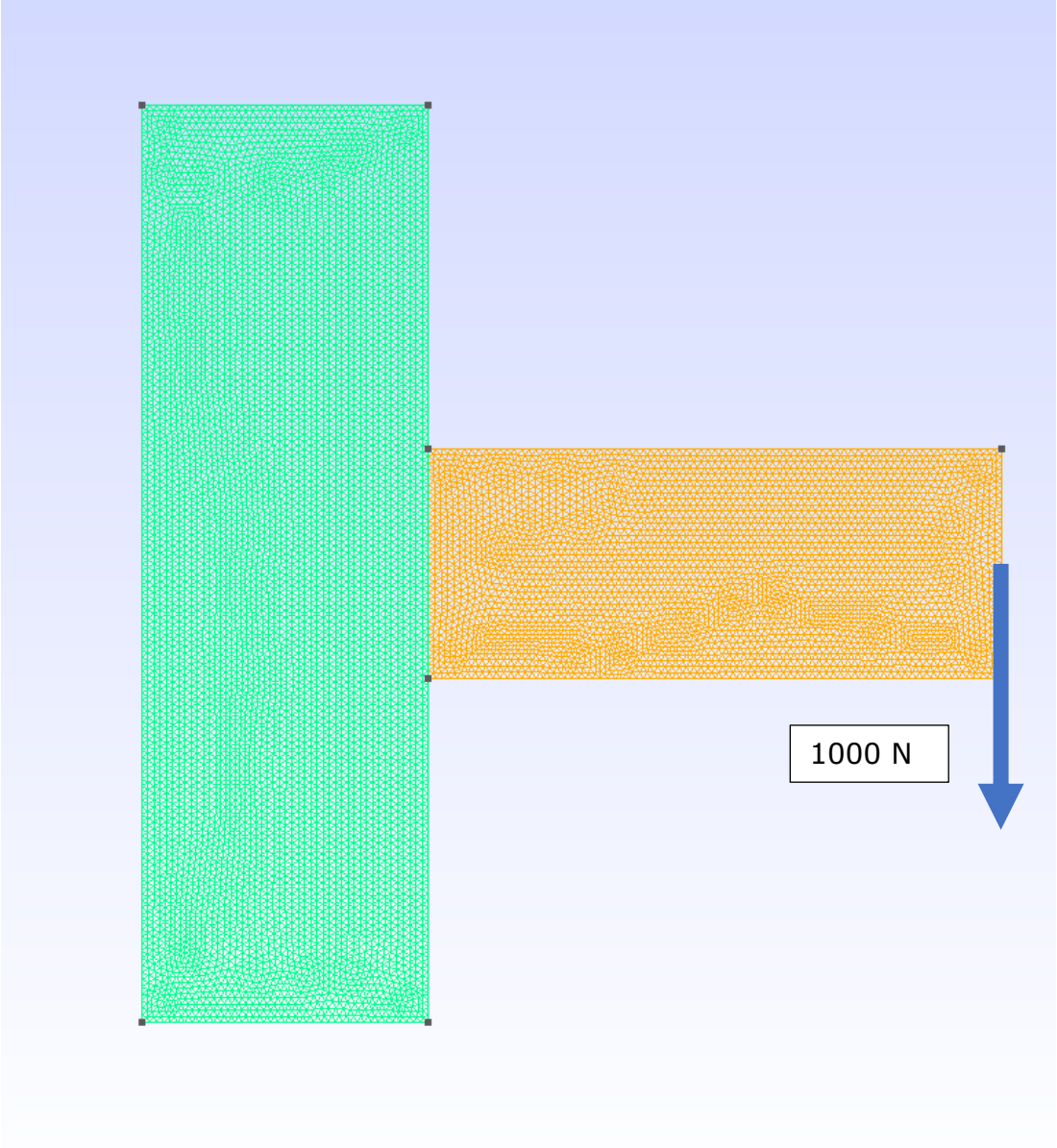

Figure 3-1 - Analysed model

It contains 18848 3-node elements that are triangles. It is loaded with 1kN of force at the end. This is a simple system of cantilever beam, but it is large enough that performing the calculations by hand is not feasible, thus it can be solved with the help of computer calculations .

The python code gets all the geometrical and material data for he analysis and constructs large stiffness matrix [K]. For this system it will be

19088x19088 matrix. This equation system was solver with the numpy module.

Then the prepared code is transferred to the virtual machine form the personal computer via SSH protocol and executed on the Virtual machine.

4. Results

The result of the analysis is a vector of the values of displacements for each of the nodes. They can be then used to obtain the deformed shape of the system and element stresses. Following pictures show the results.

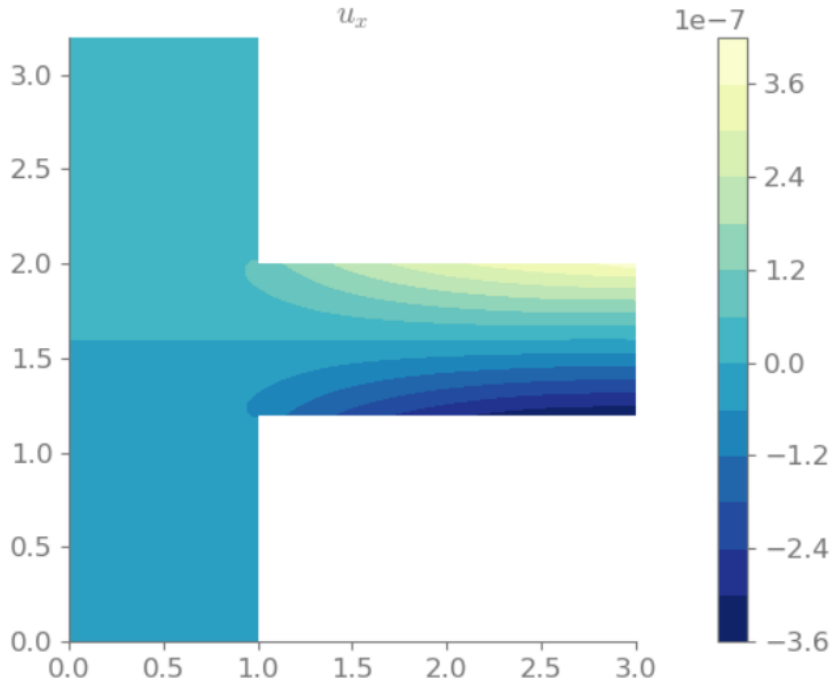

Figure 4-2 Displacement X- direction

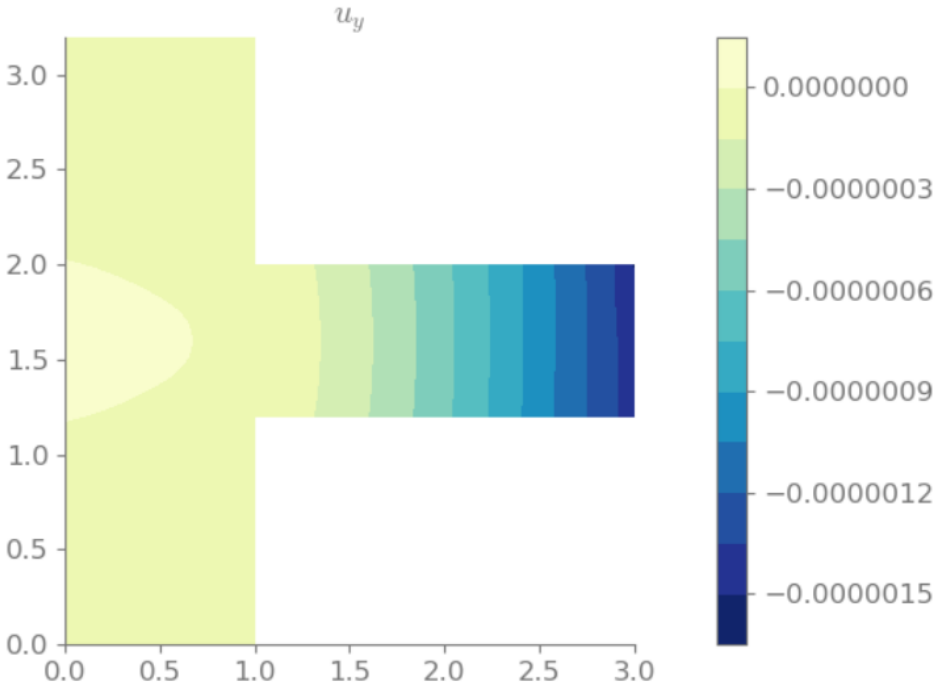

Figure 4-1 Displacement Y direction

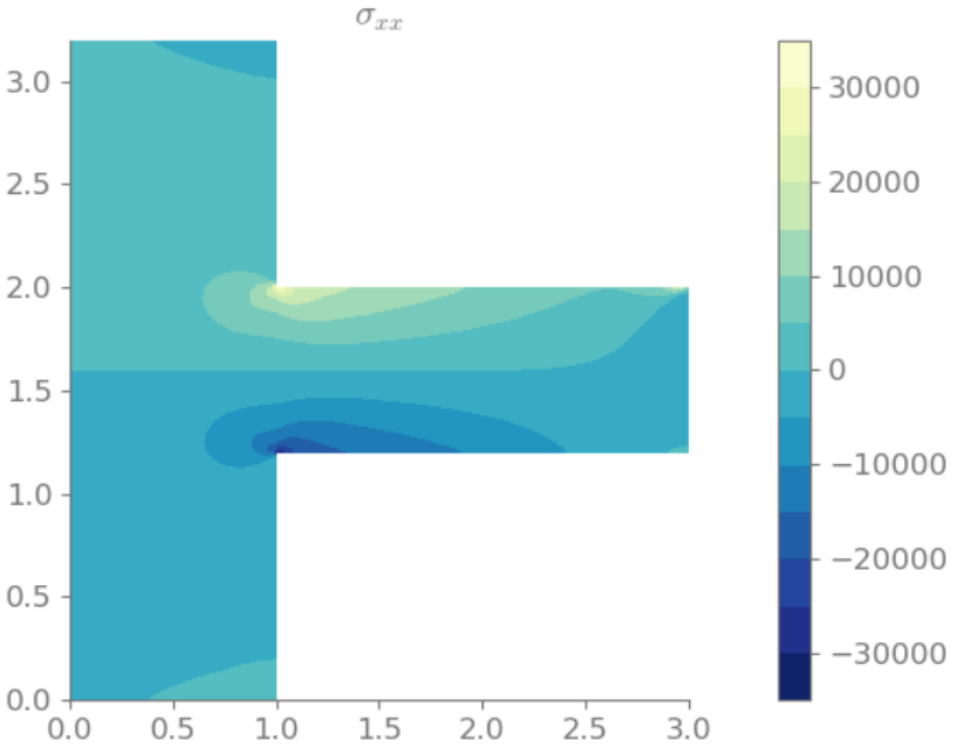

Figure 4-3 Stresses X direction

σ_{xx} stresses show that the beam is bended in the correct direction, so thus it is concluded that the simulation was set up correctly.