UDC 692:004.9

CONTEMPORARY SOFTWARE IN THERMAL ANALYSIS OF BUILDING ELEMENTS

Author – Mykhailo Zhehur[1], Stud. of gr. PCB-22-2
Scientific supervisor – Aliona Kutsenko[2], PhD, Assoc. Prof.
Language consultant – Kateryna Sokolova, PhD, Assoc. Prof.
[1]drwebspider@gmail.com, [2]kutsenko.alona@pdaba.edu.ua
Prydniprovska State Academy of Civil Engineering and Architecture

Introduction

When designing a building, it is important to ensure the future indoor environment for a comfortable life. One of the components of comfort is the temperature inside the building. To provide the optimal temperature for defined climatic conditions, a thermal engineering calculation of the enclosing structures should be carried out. With the development of technology, there is a lot of software that simplifies computations, time, and money.

This paper presents two different pieces of software for calculating building elements.

Materials and Methods

The programs under consideration don't specialize in conducting overall building analysis but in its elements. When we are talking about a whole building energy-efficiency examination, Autodesk Insight is worth mentioning. Insight allows architects and engineers to design more energy-efficient buildings due to advanced simulation engines and building performance analysis data integrated [4].

The first program is QuickField. It can perform linear and nonlinear thermal analysis for 2-D and axisymmetric models. The program is based on heat conduction equation with convection and radiation boundary conditions. Following options are available for thermal analysis:

Material properties: orthotropic materials with constant thermal conductivity, isotropic temperature dependent conductivities, temperature dependent specific heat.

Loading sources: constant and temperature dependent volume heat densities,convective and radiative sources, Joule heat sources imported from DC or ACconduction or AC or transient magnetic analysis.

Boundary conditions: prescribed temperatures, boundary heat flows, convection,radiation, and prescribed constraints for constant temperature boundaries.

Postprocessing results: temperatures, thermal gradients, heat flux densities, and totalheat losses or gains on a given part; with transient analysis: graphs and tables of timedependency of any quantity in any given point of a region.

Special features: A postprocessing calculator is available for evaluating user-defined integrals on given curves and surfaces. Plate models with varying thickness can be used for thermal analysis. The temperatures can be used for thermal stress analysis (thermo-structural coupling). Special type of inter-problem link is provided to import temperature distribution from another problem as initial state for transient thermalanalysis [3, p. 28].

COMSOL Multiphysics has a similar set of calculation capabilities to the already mentioned QuickField. COMSOL is divided into 12 modules, depending on the physical problem we need to solve.

The Heat Transfer Module is based on the three modes of heat transfer: conduction, convection, and radiation. Conduction in any material can have an isotropic or anisotropic thermal conductivity, and it may be constant or a function of temperature. Convection, the motion of fluids in heat transfer simulations, can be forced or free (natural) convection. Thermal radiation can be accounted for using surface-to-surface radiation or radiation in semi-transparent media [2].

The user should create a model, select space dimension and choose the module. After the module was selected, similar to QuickField we make geometry using COMSOL environment tools or just import ready 2D/3D models. It's necessary that physical properties were applied to the composed geometry, in our case (density, thermal conductivity, temperature).

The next significant step after creating the geometry and assigning the physics to your mode is to build the mesh. The mesh used for a model geometry plays an instrumental role in how the model is solved, as it determines factors such as:

- How the geometry is divided
- With what shape or element type the geometry is divided
- The size, density, and number of elements in the geometry
- The element quality

These factors directly affect the computation of a problem, including how long it takes a model to solve, the amount of memory required to compute a problem, how the solution is interpolated between nodes, and the accuracy of the solution [1].

When the above-mentioned procedures were completed, it's time to conduct the examination according to set conditions.

It should be noted that all calculations in specified programs are based on the finite element method.

Results

According to the results of calculations, we get almost the same value in both programs $T = 9.3497\ °C$ for QuickField and $T = 9.33\ °C$ for COMSOL. The point is selected as the coolest value on the rib that belongs to inner side of space.

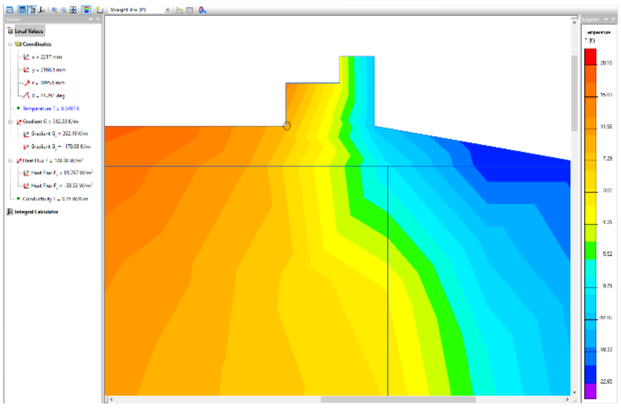

Fig. 1. Steady-state heat transfer analysis of insulated concrete wall and part of the window frame in QuickField

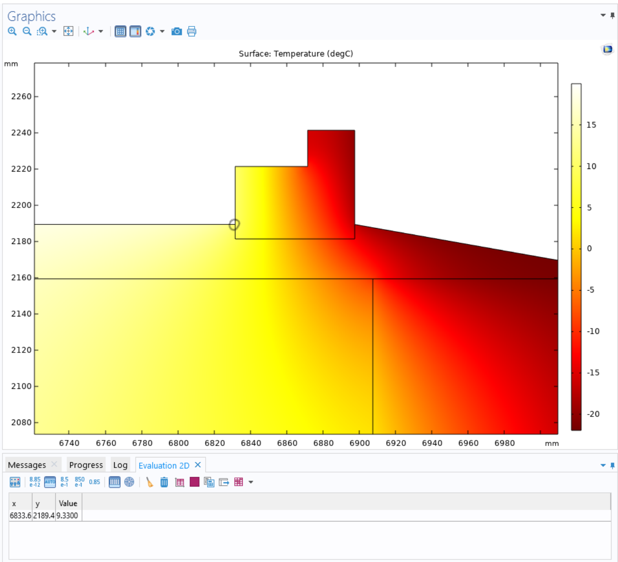

Fig. 2. Steady-state heat transfer analysis of insulated concrete wall and part of the window frame in COMSOL Multiphysics

Conclusions

The approaches described can contribute to understanding the logic which is the basis of this kind of software. An ability to use FEM programs for heat analysis helps use resources more rationally and perform a more detailed review.