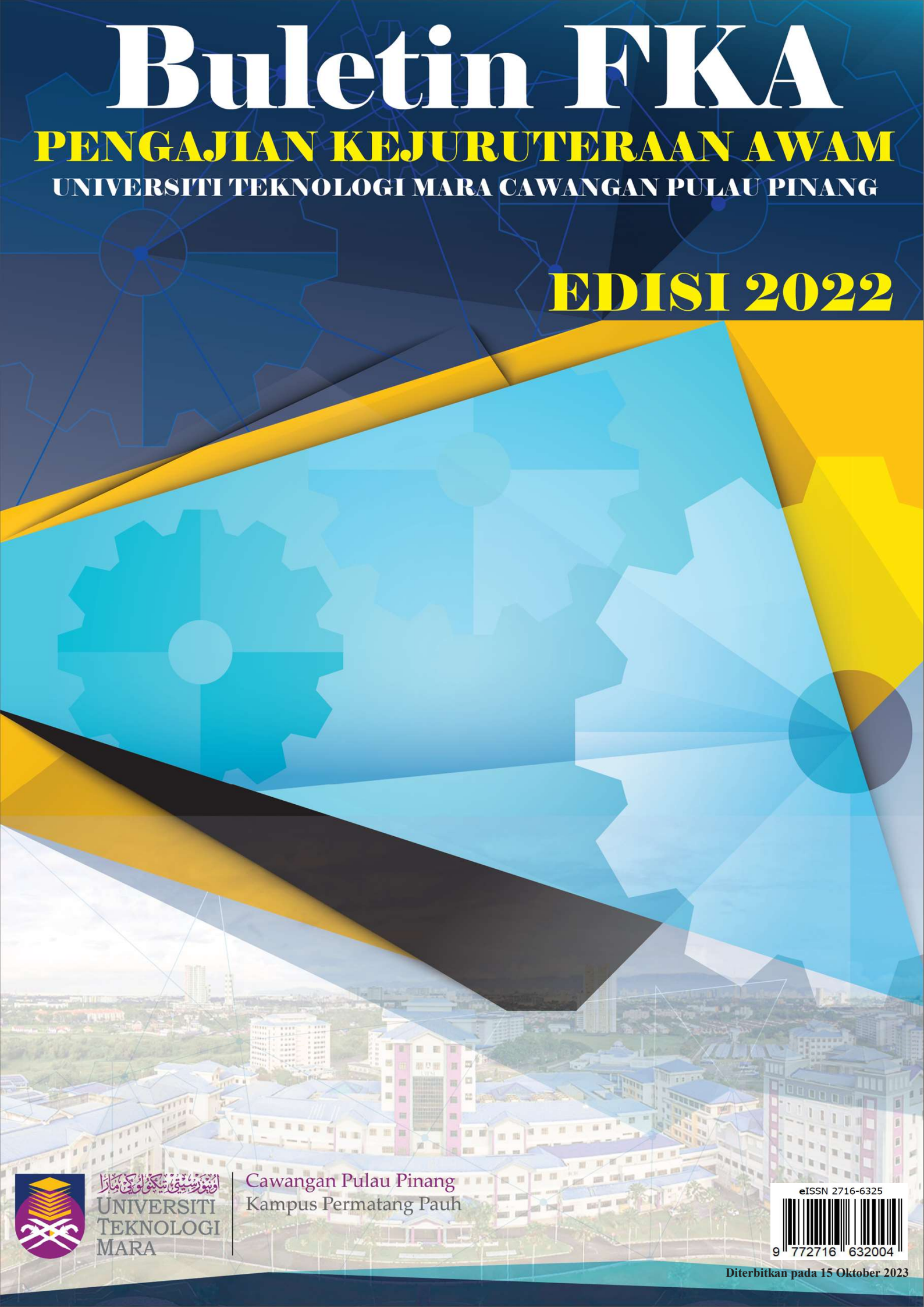
Buletin FKA

PENGAJIAN KEJURUTERAAN AWAM

UNIVERSITI TEKNOLOGI MARA CAWANGAN PULAU PINANG

EDISI 2022



UNIVERSITI TEKNOLOGI MARA

Cawangan Pulau Pinang
Kampus Permatang Pauh

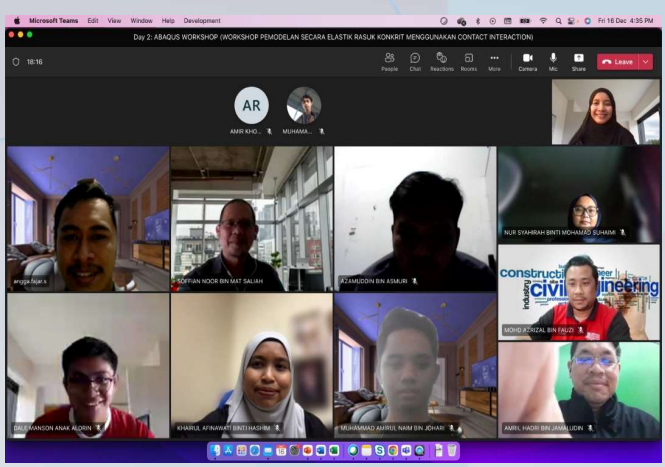
eISSN 2716-6325

Diterbitkan pada 15 Oktober 2023

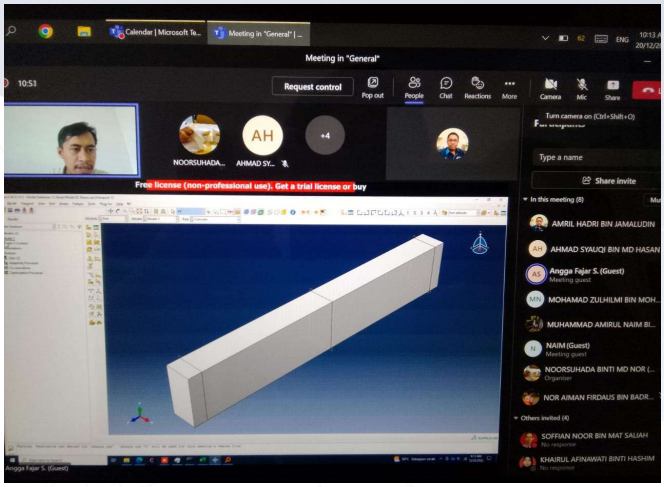
Workshop on Finite Element Method (FEM) for Final Year Project Student and Supervisors

By: Noorsuhada Md Nor, Soffian Noor Mat Saliah, Ahmad Syauqi Md Hasan, Amril Hadri Jamaludin, Mohd Azrizal Fauzi, Amir Khomeiny Ruslan, Khairul Afinawati

The cheerful and prepared faces of Dr. Soffian and Dr. Angga at the opening of the workshop



The happiness and seriousness on the faces of the workshop participants



Dr. Angga explained FEM on a sample beam

A workshop on Finite Element Method (FEM) was held virtually on Microsoft Teams from 9th to 23rd December 2022. The workshop was led by Noorsuhada Binti Md Nor and attended by 21 final year project students, supervisors and staff from UiTM Cawangan Pulau Pinang. The workshop was conducted in collaboration with expert speaker Dr. Angga Fajar Setiawan from Universitas Gadjah Mada.

The first day of the workshop, 9th December 2022, focused on "ABAQUS for Analysis". The participants were given an introduction to FEM concepts and available tools in ABAQUS. However, due to the high cost of ABAQUS, the participants used The ABAQUS Learning Edition, which is available free of charge and supports structural models up to 1000 nodes on Windows platform only.

The subsequent days of the workshop, held on 12th, 16th, 19th, and 23rd December 2022, focused on "Elastic Modelling of Concrete Structures using Contact Interaction", "Inelastic Modelling of Concrete Structures containing Single Reinforcement", "Inelastic Modelling of Beams containing Doubly Reinforced Steel", and "Analysis of Simulation Results using Abaqus", respectively.

The workshop highlighted the benefits of ABAQUS FEM in engineering analysis, including its ability to solve complex problems such as dynamic, thermal, and structural stability analyses. ABAQUS FEM is a flexible and user-friendly software that enables users to create complex models quickly and efficiently. The software can model various types of materials, including plastics, metals, composites, and elastomeric materials. It provides high-quality and accurate analysis results that can be trusted by users.

The workshop concluded with positive feedback from participants. One participant expressed their satisfaction with the workshop, stating that it helped them achieve their final year project objective of completing their wall panel model. The workshop was short, informative, and beneficial to the participants. The hope is that by conducting this short course, it will encourage more curiosity and understanding of modelling concepts among future students in Malaysia.

In conclusion, the workshop was a success, and the author hopes that this series will continue in the future, involving more lecturers and students who are interested in expanding their knowledge beyond what is taught in university classes. The author extends their gratitude to all those involved in the workshop.