# One Page Summary

Finite Element Analysis (FEA) has emerged as a reliable analysis approach as a result of the advancement of computational methods and power. Nevertheless, accurate representation of large-scale problems using FEA might be hindered by the impractical computational demand, particularly for problems requiring extensive iterations or multiple run times. One of these cases is when deriving fragility curves where often nonlinear dynamic analysis is needed, requiring extensive number of simulations. Previous studies have addressed the issue of the high computational cost for developing fragility functions by reducing the resolution of the model, use static based procedures or use response surface methodology and deep neural networks to establish a relation between inputs and output. These mitigation measures can yield a significant impact on the accuracy of the derived fragility curves. In this study, a two-stage computationally efficient procedure is proposed to develop fragility curves without compromising on accuracy. specifically, two models are developed one of them is fully detailed and the other is simplified with some parameters that can be adjusted if needed. The two models are analyzed and integrated where the simplified model parameters are updated so that both models' results are as close as possible to each other. Once the simplified model is calibrated to the detailed model, fragility curves can be derived using the simplified model in a much more efficient manner.

To calibrate the simplified model, a genetic algorithm-based optimization technique is used as the calibration tool to update the simplified model based on the difference between the pushover curves between the detailed and simplified models. Then the updated model is used to derive accurate fragility curves. In order to evaluate the proposed procedure, non-linear dynamic analysis is used to derive fragility curves for a four-story buckling restrained braced frame hospital designed in a study by NIST in Memphis, Tennessee. The detailed model is developed using ANSYS WORKBENCH, which is a general-purpose finite element software, utilizing 3D solid meshes. The simplified model, on the other hand, is developed using ANSYS APDL utilizing beam elements. The updating procedure is applied to the simplified model using the genetic algorithm to find the most optimal parameters for the model. Finally, the updated model is used to develop fragility curves aiming to capture the behavior of the detailed model as close as possible. The results showed the proposed framework was able to increase the accuracy of the fragility function by more than 20% at some values of the spectral acceleration. The general framework is shown in the following figure.