FE ANALYSIS ON ELECTRIC MOTOR SUBJECTED TO EARTHQUAKE

[1]Sergio Macchiavello[*], [1]Alessandro Bozzolo, [1]Claudio Brunetto, [2]Massimiliano Di Chiara
[1]D'Appolonia S.p.A., Italy
[2]Nidec ASI S.p.A., Italy

KEYWORDS –
Electric Motor, Response Spectrum Analysis, Finite Element Method

ABSTRACT –
A Finite Element approach has been applied to assess the structural response of an industrial electric motor designed and produced by Nidec ASI S.p.A. subjected to earthquake, in order to classify its performances according to the seismic safety groups provided by standards.
The electric motor was a 4 poles induction motor CR 710 Y 4, for variable speed industrial application with a rated power of 2500kW.

The 3D discretized models of the following components of the electric motor were generated: motor housing, shaft, front and rear shields; the other components were considered into the analyses through simplified models.

The 3-D CAD models of the abovementioned components were simplified and discretized using ANSA pre-processing tool. The same tool was also used to generate beam elements, lumped masses, simplified bearings elements, contacts and other connections among components. The model was hence imported in one of the most common commercial FEM software. The followed Finite Element approach foresees the execution of static, pre-stressed modal and single-point response spectrum (SPRP) analyses.

Loads related to the working conditions are: gravity acceleration, electromagnetic force between stator and rotor, rotational velocity, shaft torque and rotor unbalance force.

The seismic load is the earthquake response spectrum, in terms of acceleration vs. frequency, calculated, considering an initial damping of 2%.

Calculated stresses and displacements allowed to attribute the electric motor to the proper seismic safety group according to standards.