FEMSA: A Finite Element Simulation Tool for Quasi-static Seismic Deformation

Manuela Votpe, Daniele Melini, Antonio Piersanti
votpe@ingv.it, melini@ingv.it, piersam@ingv.it

INTRODUCTION

Modeling stress and displacement fields resulting from seismic faulting is an increasingly valuable tool in earthquake studies. In the last few decades, many different approaches have been proposed to study the static and quasi-static deformations associated with a seismic event.

On the one hand, analytical and semi-analytical models, although often very accurate and simple to implement, suffer from a limited domain, as their use very often opportunely to account for geodynamic and/or tectonic complexities. On the other hand, pure numerical models have been developed to overcome restrictions and improve performances with respect to analytical techniques, but in some cases they also result unable to model at the optimal level features or geometric complexities of interest. Among numerical methods, however, FEMSA: A Finite Element tool, modeling three-dimensional flexible modeling, is less affected by the absence of any user-code interaction. Furthermore, in fact involves other analytical disadvantages since the test both remain accessible to the whole scientific community for different reasons. Commercial codes have often prohibitive prices and are typically affected by the absence of any user-code interaction. Moreover, most numerical codes are aimed to industrial design problems and their adaptation to geophysical problems is not straightforward. In-house codes, in most of the cases, turn out to be both understandable and usable just to develop their design and are often designed to specific problems, resulting hardly appealable to more general cases. Furthermore, they are not particularly robust with respect to errors or inaccuracies, as they are instead based on ad-hoc made FEM models that are actually not well understood and tested.

What about different boundary conditions?

Example: bottom and lateral nodes kept fixed in the direction perpendicular to the surface

Example: bottom and lateral nodes kept fixed in the direction parallel to the surface

** in the next future con...

* this choice is not mandatory: other element types can be implemented

** in the next future contact elements will be available

BENCHMARK SIMULATIONS

Test plane domain 100 x 200 x 200

Hypermesh domain with user-code mesh "Okada boundary conditions"

Picture galleries:

a) Vector plot for the ground horizontal displacement field: numerical (d) vs analytical (e) solution

dip=90°

right lateral strike slip

strike=30°

dip=60°

dip slip

normal

strike=0°

dip=20°

dip=90°

dip=60°

dip=20°

dip=90°

dip=60°

dip=20°

dip=90°

dip=60°

dip=20°

dip=90°

dip=60°

dip=20°

right lateral strike slip

strike=30°

dip=60°

dip=20°

dip=90°

right lateral strike slip

strike=30°

dip=60°

dip=20°

dip=90°

right lateral strike slip

strike=30°

dip=60°

dip=20°

dip=90°

right lateral strike slip

strike=30°

dip=60°

dip=20°

dip=90°

right lateral strike slip

strike=30°

dip=60°

dip=20°

dip=90°

right lateral strike slip