Simulation of the propagation of cracking by finite element code calculation of a supermartensitic stainless steel pipe

K.SLIMANI1, 2, M.HASSANI 2, S. TLILI1, 3, C.E. RAMOUL1, T. HAZEM1

Abstract : The biggest concern of industrial manufacturers is the sustainability of their equipment and facilities. In the oil sector, for example, cylindrical pipe for the energy transportation to suffer corrosion damage or corrosion stress cracking is a complex problem since it requires to follow the geometry of the crack over time. Many numerical methods are applied to the propagation of fatigue cracks that require conditions to correct limitations in loading term to get results close to reality.The method of Finite Elements is a way to simulate crack propagation. However, it presents two major difficulties, the first is the use of a very fine mesh around the crack tip and the second difficulty is the explicit representation of the crack that the problem of cracking path during the simulation of propagation. Indeed, let the mesh is constructed with a priori knowledge of the route or it is changed every time the crack advance.In this context, we studied law cracking of internal pressure pipe of super martensitic stainless steel 13% Cr and 5% Ni 2% Mo by a simulation using a finite element code "ANSYS " with a calculation of the stress intensity factor and prediction of the lifetime.

Keywords : simulation, Finite Elements, Crack, Pipe, Super Martensitic Stainless Steel.