Abstract: The main goal here is to optimise the finite element mesh used to predict plasticity induced crack closure (PICC). A numerical model was developed for a M(T) specimen made of 6016-T4 aluminium alloy. The parameters studied were the size of most refined region perpendicularly to crack flank ($y_m$) and along propagation direction ($x_r$), the size of finite elements near crack tip ($L_1$) and the vertical size of refinement close to crack flank ($y_{A/B}$). A maximum size of about 1.3 mm was found for $y_m$, but a smaller value has a limited impact on PICC. An analytical expression was proposed for $x_r$, dependent on $D_K$ and $K_{max}$. An optimum value seems to exist for $L_1$. 