Associazione EURATOM ENEA sulla Fusione

Multipurpose ANSYS FE procedure for welding processes simulation

Andrea Capriccioli, Paolo Frosi – ENEA CR Frascati, Via Enrico Fermi 45, 00044 Frascati Italy

Foreword

This work deals with the development of numerical models related to thermal and mechanical simulation of welding process. With the use of ANSYS code, we have been able to consider parameters like the welding speed, the number and sequence of passes, the filling material supplying, and also the geometrical constraints, the material nonlinearities, the convection and irradiation phenomena.

The thermal and mechanical material properties of both INCONEL 625 and AISI 316 are described till vapour phase; the 3D ANSYS model use both brick and non-linear contact elements, and also the birth and death procedure has been used extensively. Moreover we assume that the displacements of the parts, during the welding, don't affect the thermal distribution of the parts themselves. So we can carry on an uncoupled analysis.

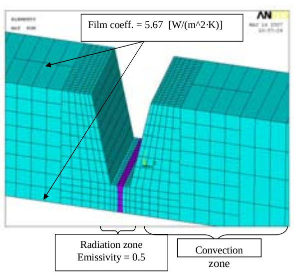
Before reaching the final model we have performed several sensitivity analysis changing mesh size, welding speed and material properties to understand the effect of these parameters on overall results (the joint's thermal shrinkage and the residual stress field).

Several aspects of welding modeling that have been studied are common to Laser and TIG method, but we report a full simulation of a TIG specimen (source W7-X) having the root seam plus 14 passes with a non constant gap: the thermal and mechanical results are presented.

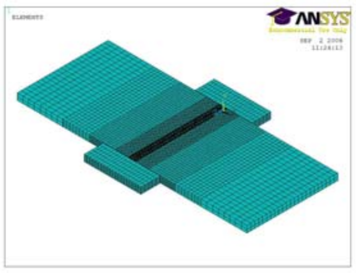
Thermal analysis

In fig.1 we have an example of a model of laser welding of two Inconel 625 plates: the zone near the caulker has a finer mesh while the rear zone has a coarser one; they are tied together with the contact element (MPC option).

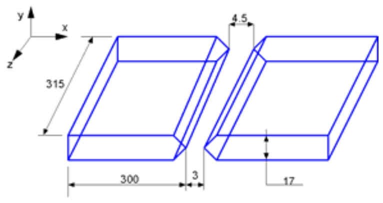
In fig. 2 we have the ANSYS model of the TIG welding specimen of AISI 316 with the lateral restraints (W7-X): the dimensions of test piece are reported in fig.3; in fig.4 there is the scheme for subsequent passes.



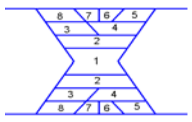
(fig.1)



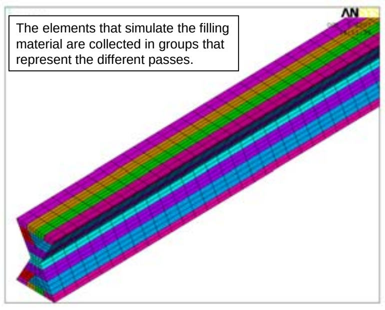
(fig.2)



(fig.3)



(fig.4)



Sensitivity analysis: welding speed

With the same mesh size, we have varied the welding speed ranging from 0.25 to 10 mm/s and we have obviously found that temperature gradients increase with speed.

In fig.5 we find the highest temperatures of the root seam at different welding speed; the power increases with speed because of the shorter application time (power/speed: 2.5/600; 5/800; 10/1150)

Sensitivity analysis: mesh size

With the same welding speed (2.5 mm/s), we have varied the mesh size ranging from 10 to 50 mm in element length and we have verified that the thermal distributions at the same time are very similar.

In fig.6 we have the highest temperatures of the root seam with different element length and with constant welding speed; the last part of the curves are related to the cooling time.

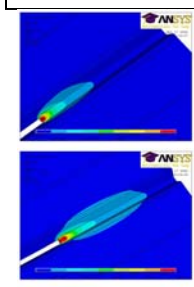
Sensitivity analysis: material property

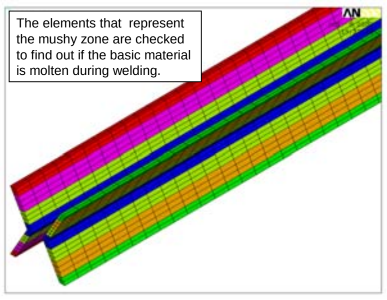
The specific heat in liquid phase has been changed of ± 10%: the highest temperature has been affected by a variation of only 1K.

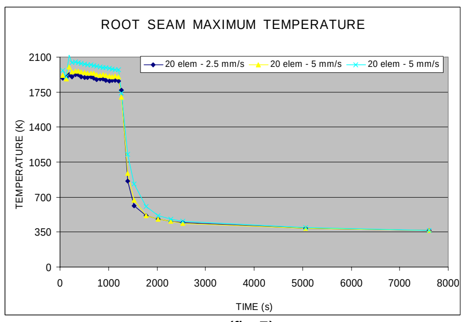
The thermal conductivity has been changed of ± 10% and we have observed no evident differences between the reference case and the others.

In the subsequent mechanical analysis the largest variation in maximum displacement has been of very few thousandths of mm.

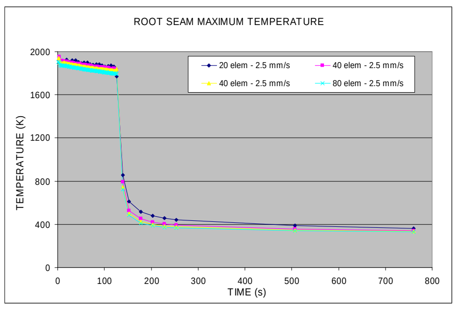
Sensitivity analysis: size of melted "drop"



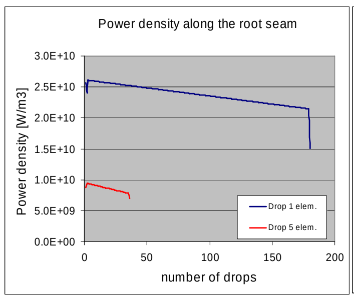


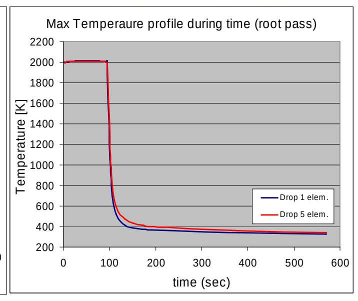


(fig.5)



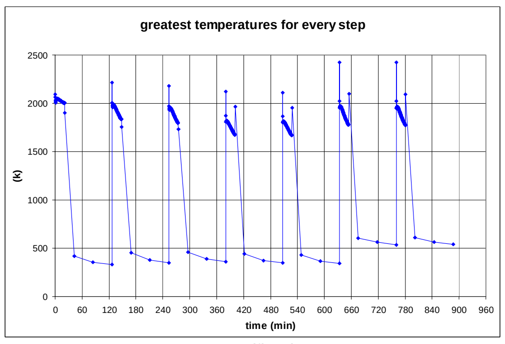
(fig.6)



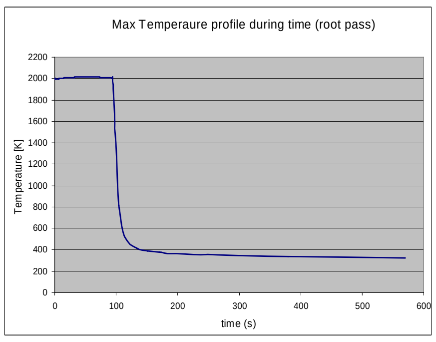


Remarks on thermal power density

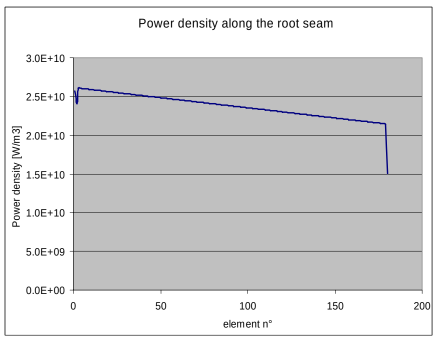
In the previous cases we have applied a *constant power* but the maximum temperature is not constant because the element size increases along the axis (the gap size is variable). We find the same behavior in the subsequent passes (fig.7); and also we find that the absolute temperatures and the downward slope between different passes are not the same: this is because the passes have their own position, shape and boundary conditions. If we applied a *constant power density*, the maximum temperatures increases along the axis because the gap and the volume of the single "drop" increase but the contact surface with the basic material remains constant. For this reason we have developed a fast iterative procedure that allow to evaluate the power density needed to reach an assigned temperature. With this system the maximum temperature remains constant for the entire pass and for all elements (fig. 8). The power density that must be applied in general is not constant (fig.9). Finally we have performed the same analysis (fast iterative procedure) for every pass and we have found a different *"power density history"* for each pass to obtain the same maximum temperature (fig.10).



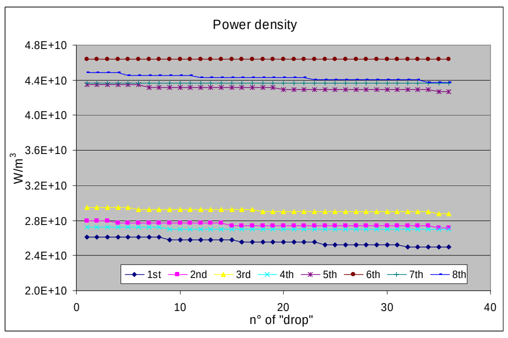
(fig.7)



(fig.8)



(fig.9)



(fig.10)

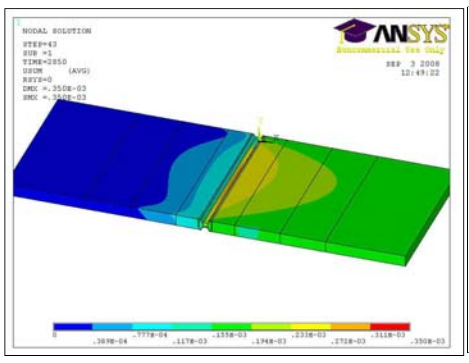
Mechanical Analysis

For thermal analysis the computer time is about 20h/pass/m with 1 mm element length (Pentium 4 CPU 3GHz, 2Gb RAM) and we have had no problem with convergence. Otherwise, in mechanical analysis we have had several convergence problems : so we have had to use the shorter element as possible (1.3 mm). In this case the computer time is 4-5 time longer than the thermal one; to overcome this difficulty we have developed a special procedure to obtain a conservative solution.

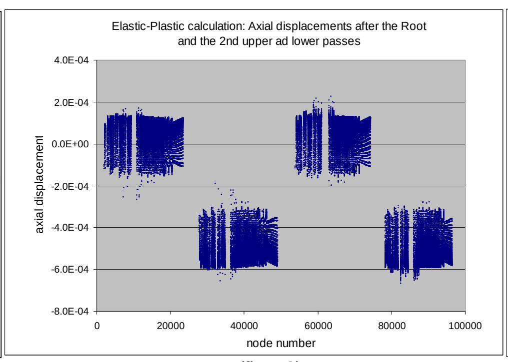
At first we have performed the entire mechanical analysis, then we have stored the nodal displacements in the shrinkage direction of the filling material and the fine mesh region very near to the caulker. Practically we have two sets: the first one is relative to the nodal displacements of the plate that is clamped; they result in a band around zero value. The second one is relative to the nodal displacements of filling material and of the other plate that, at this point, really represent the shrinkage. In fig. 11-12 there is a contour plot and a graph relative to the total displacement after the second upper and lower passes; for the other passes the graphs are similar. These set of result is a sort of reference case.

After that we have performed a distinct mechanical analysis for every pass (we will call this case "pseudo-linear approach"): the first one is related to the root pass and is equal to that of the previous case; the other analysis are related to the corresponding other passes but they have been executed beginning from the configuration of their own previous pass (with the respective elements reactivated), but with initial zero stress and strain. The respective average values of shrinkage displacement for each pass is grater than that obtained in the real case because the material starts from a free-stress state and it has no deformation energy accumulated. In fig. 13 we plot the subsequent average displacements for all passes in the total elastic-plastic approach and pseudo-linear one.

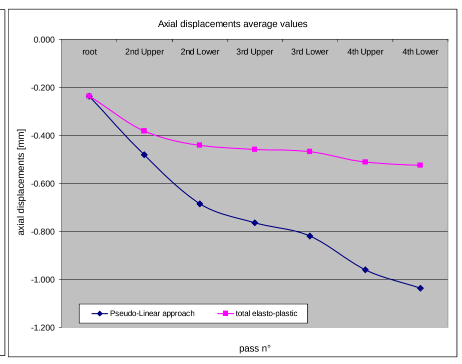
In fig.14 we plot the contribution of every pass in the same two cases: we observe that the greater contribution to the global displacement is given in the early passes; then when the caulker is going to fill, the resistant section increases and the additional thermal loads does not add further great strains. We can see immediately that "pseudo-linear" approach is conservative: it allows to overestimate the final displacements. So the pseudo-linear approach can be used to reduce the calculation time: indeed it is possible (after performing the "ordinary" thermal analysis) to execute at the same time a set of mechanical analysis as much as the number of passes and then simply add the contribution of each pass. The next pass of the analysis is to think about a procedure that, on the basis of the results obtained with the two approaches, could reduce the overestimation involved in the simplified analysis. In this way it is possible to run the whole welding simulation for a long joint with a considerable saving in computer time.



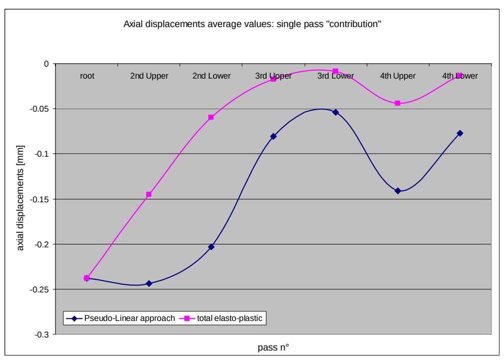
(fig.11)



(fig. 12)



(fig. 13)



(fig. 14)