COMPUTER MODELLING OF HONEYCOMB SANDWICH PANELS

For a more sophisticated analysis of a structure, considering the sandwich panel to be subjected to a combination of forces, a technique such as Finite Element Analysis (FEA) might be used.

In general terms, the shear forces normal to the panel will be carried by the honeycomb core. Bending moments and in-plane forces on the panel will be carried as membrane forces in the facing skins.

For many practical cases, where the span of the panel is large compared to its thickness, the shear deflection will be negligible. In these cases, it may be possible to obtain reasonable results by modelling the structure using composite shell elements. It should be noted that the in-plane stiffness of the honeycomb is negligible compared to that of the facing skins.

Where a more detailed model is required it is possible to model the honeycomb core using solid 3D elements. Attempts to model the individual cells of the honeycomb should be avoided for normal engineering analyses.

When defining the properties of honeycomb core the following points should be taken into consideration:-

$E_X \approx E_Y \approx 0$

A very small value may be necessary to avoid singularity.

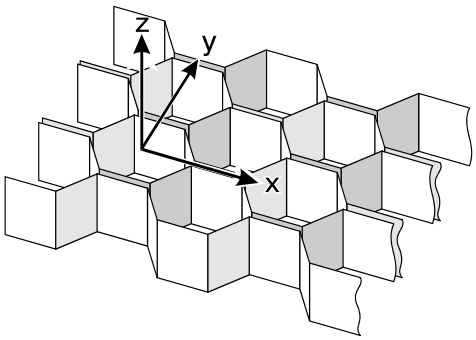
$\mu_{xy} \approx \mu_{xz} \approx \mu_{yz} \approx 0$

$G_{xy} \approx 0$

$G_{xz} = G_L$ = shear modulus in ribbon direction

$G_{yz} = G_W$ = shear modulus in transverse direction

$E_Z = E_C$ = compressive modulus of core material



Before analysing a large structure the modelling technique should be checked by modelling a simple panel with known results.

The above simplistic approach has proven to give reasonable engineering solutions for practical applications.

The actual force/stress distribution within a honeycomb sandwich structure is a complex subject, and is beyond the scope of this publication.