The heat transfer mechanism in friction stir welding (FSW) is not fully understood. This study attempts to establish a congruent approach towards estimating the heat source in friction stir welding and study its impact on the thermal profiles and histories numerically using Finite Element Method. To eliminate the uncertain frictional coefficient associated with conventional friction based heat transfer model, which is also less satisfactory in capturing the deformational heating in FSW, an input-torque based heat generation model is proposed. The moving heat source engendered by the rotation and linear traverse of the tool is correlated to the actual machine power input which is apportioned to the different interfaces the tool makes with the weldpiece based on the torques at different tool surfaces. This novel approach is formulated in a three dimensional finite element model and implemented for different materials--AA 6061-T6, AA 7050, AA 2024 and SS 304L—to successfully predict the temperature distributions during FSW of those metals. Excellent agreement between the simulated temperature profiles and experimental data demonstrates the functionality of the model. The effects of various heat transfer conditions at the bottom surface of the workpiece, thermal contact conductances at the work-piece and the backing plate interface, and different backing plate materials on the thermal profile in the weld material are also investigated computationally. The heat transfer condition at the bottom of the welding plates is found to be the most important external factor dominating the thermal cycles.

It is also found that incorporating the convective material transport within the plastically deformed zone gives rise to an asymmetric temperature distribution with the advancing side experiencing slightly higher temperatures than the retreating side.

An attempt has been made to predict the thermal residual stress distributions in friction stir welded workpieces by sequentially coupling the thermal histories developed by the input torque based model into a mechanical model assuming elastic-perfectly plastic metal behavior in accordance with the classical metal plasticity theory. The longitudinal stress components are found to be the highest tensile stress components and correspond to the temperature profiles within the heat affected zone of the weld. The through-thickness (normal) stresses are found to be negligible compared with the longitudinal and transverse stress components.

To facilitate simulation runs of the proposed models, a software package was also developed using the programming language C that allows users not familiar with Abaqus pre- or post-processors to successfully create appropriate Abaqus input files and extract the thermal history from the subsequent outputs.