

عنوان مقاله:

Analysis Of Stress In Friction Stir Welding Process Of Aluminum-Steel Sheets With Finite Element Using Abaqus Software

محل انتشار:

دوازدهمین کنفرانس بین المللی راهکارهای نوین در مهندسی ، علوم اطلاعات و فناوری در قرن پیش رو (سال: 1401)

تعداد صفحات اصل مقاله: 16

نویسنده:

.Amirhossein Fardadfar - Department of Mechanical Engineering, Tiran Branch, Islamic Azad University, Tiran, Iran

خلاصه مقاله:

Welding of aluminium-steel sheets is relatively difficult, mainly due to the high thermal conductivity of aluminium and the formation of defects such as porosity and freezing cracks during welding. In shipyards that use aluminium in the manufacturing process, welding is often done manually by welding Arc welding is performed under friction stir welding is a relatively new bonding process that has been proven to bond various types of metals such as steel, titanium, lead, copper and aluminium. This process is especially common for bonding aluminium and steel. It is profitable and cost-effective, and many industries around the world use it commercially. The use of this welding method has proven advances in the cost of manufacturing the life of joints and the distortions caused by welding. In this paper, after examining the effective parameters in this welding method, the finite element analysis of the friction perturbation .welding process is performed and the stress contours resulting from this operation are investigated and presented

کلمات کلیدی:

Friction Stir Welding, Aluminum-Steel Sheets, Finite Element, Stress Analysis, ABAQUS Soft Ware

لینک ثابت مقاله در پایگاه سیویلیکا:

<https://civilica.com/doc/1483737>

