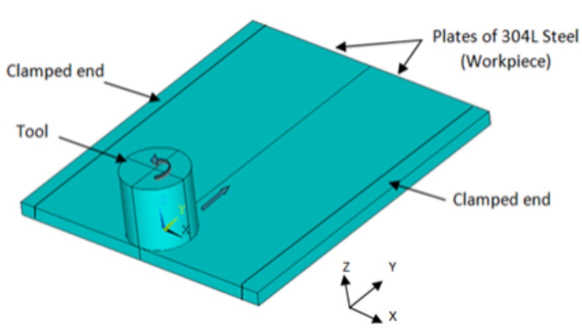
Aim: Perform reverse engineering of Friction stir welding process.

Abstract:
3D printing technology is making its mark in automotive, aerospace, and bio-medical-related industries. It is considered a viable option for the direct manufacturing of final parts. However, it is not possible to print longer parts in a single attempt due to the bed size limitation of printers. This problem can be addressed by employing a polymer joining technique as a secondary operation. Moreover, low mechanical strength and inferior geometrical qualities like the flatness of the joined parts restrict its real-time industrial application. Here, an attempt is made to join a longer part (typical of an aircraft wing) using friction stir welding technique. Joining was performed on 3D printed similar/dissimilar thermoplastic parts.

Introduction:
Friction stir welding (FSW) is a solid-state welding technique that involves the joining of metals without filler materials. A cylindrical rotating tool plunges into a rigidly clamped workpiece and moves along the joint to be welded. As the tool translates along the joint, heat is generated by friction between the tool shoulder and the workpiece. Additional heat is generated by plastic deformation of the workpiece material. The generated heat results in thermal softening of the workpiece material. The translation of the tool causes the softened workpiece material to flow from the front to the back of the tool where it consolidates. As cooling occurs, a solid continuous joint between the two plates is formed. No melting occurs during the process, and the resulting temperature remains below the solidus temperature of the metals being joined. FSW offers many advantages over conventional welding techniques, and has been successfully applied in the aerospace, automobile, and shipbuilding industries.

Problem Description:
A direct coupled-field analysis is performed on the given model.
The simulation welds two 304L stainless steel plates (workpiece) with a cylindrical shape tool, as shown in the following figure:

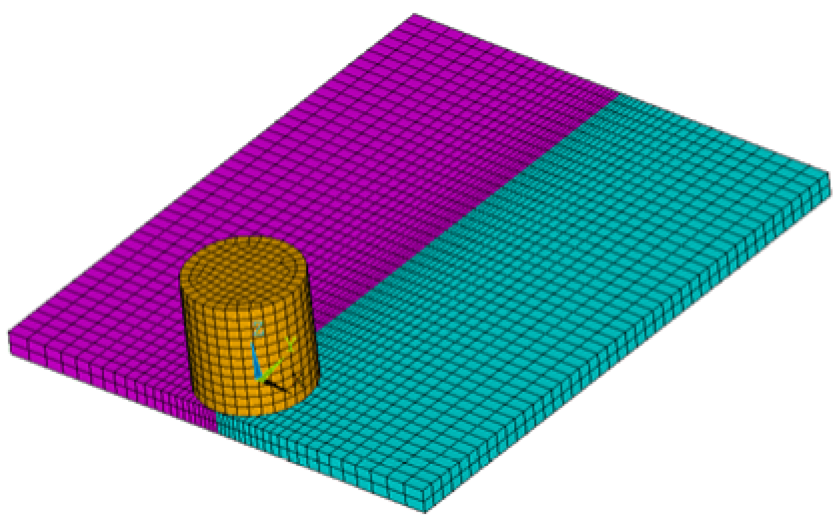


A cylindrical PCBN tool is modeled in this case.
The simulation is performed in three load steps, each representing a respective phase (plunge, dwell and traverse) of the FSW process.

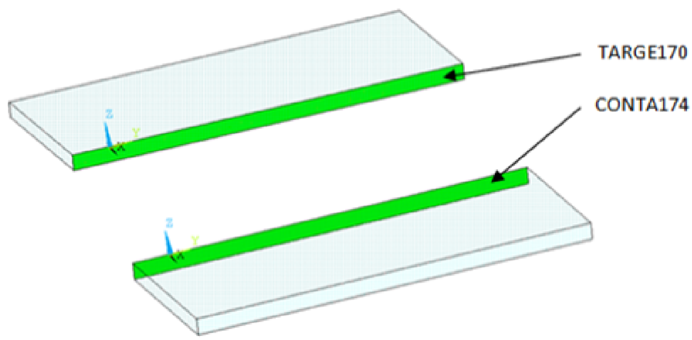
<u>Modeling:</u>

Modeling is a two-part task, as described in these topics:

1. **Workpiece and Tool Modeling**
   Two rectangular shaped plates (similar to those used in the reference model) are used as the workpiece. Dimensions have been reduced to decrease the simulation time. The plate size is 3 x 1.25 x 0.125 in (76.2 x 31.75 x 3.18 mm). The tool shoulder diameter is 0.6 in (15.24 mm).
   A hexahedral mesh is used instead of a tetrahedral mesh to avoid mesh-orientation dependency. For more accurate results, a finer mesh is used in the weld-line region. The following figure shows the 3-D meshed model:

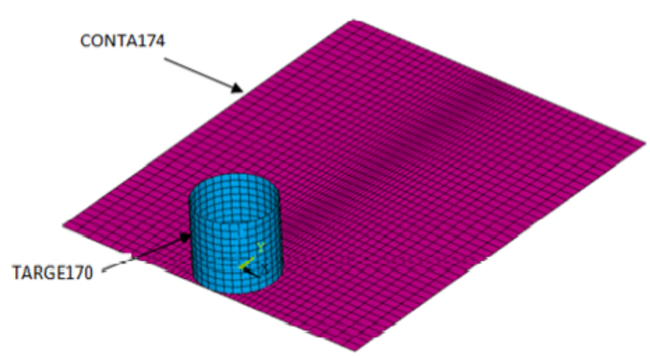


2. **Contact Modeling:**
   Contact is modeled as follows for the FSW simulation:
   I. **<u>Contact Pair Between the Plates</u>**:
      During the simulation, the surfaces to be joined come into contact. A standard surface-to-surface contact pair using TARGE170 and CONTA174, as shown in the following figure:



The surface-projection-based contact method (KEYOPT(4) = 3 for contact elements) is defined at the contact interface. The surface-projection-based contact method is well suited to highly nonlinear problems that include geometrical, material, and contact nonlinearities.
To achieve continuous bonding and simulate a perfect thermal contact between the plates, a high thermal contact conductance (TCC) of 2E06 W/m2 °C is specified.
Welding occurs after the temperature of the material around the contacting surfaces exceeds the bonding temperature (approximately 70 percent of the workpiece melting temperature).

II. **<u>Contact Pair Between Tool and Workpiece:</u>**
The tool plunges into the work piece, rotates, and moves along the weld line. Because the frictional contact between the tool and workpiece is primarily responsible for heat generation, a standard surface to surface contact pair is defined between the tool and workpiece. The CONTA174 element is used to model the contact surface on the top surface of the workpiece, and the TARGE170 element is used for the tool, as shown in this figure:



The factor for the distribution of heat between contact and target surfaces is defined next; the FWGT real constant is set to 0.95, so that 95 percent of the heat generated from the friction flows into the workpiece and only five percent flows into the tool.

A low TCC value (10 W/m2 °C) is specified for this contact pair because most of the heat generated transfers to the workpiece. Some additional heat is also generated by plastic deformation of the workpiece material. Because the workpiece material softens and the value of friction coefficient drops as the temperature increases , a variable coefficient of friction (0.4 to 0.2) is defined (TB,FRIC with TBTEMP and TBDATA).

III. Rigid Surface Constraint:

The workpiece remains fixed in all stages of the simulation. The tool rotates and moves along the weld line. A pilot node is created at the center of the top surface of the tool in order to apply the rotation and translation on the tool. The motion of the pilot node controls the motion of the entire tool. A rigid surface constraint is defined between the pilot node (TARGE170) and the nodes of the top surface of the tool (CONTA174). A multipoint constraint (MPC) algorithm with contact surface behavior defined as bonded always is used to constrain the contact nodes to the rigid body motion defined by the pilot node.

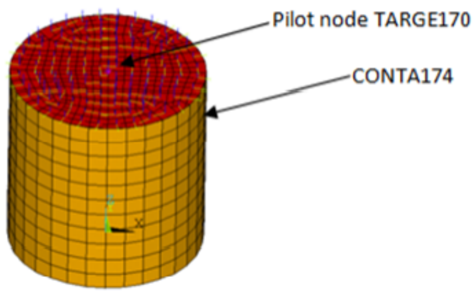
The following contact settings are used for the CONTA174 elements:

- To include MPC contact algorithm: KEYOPT(2) = 2
- For a rigid surface constraint: KEYOPT(4) = 2
- To set the behavior of contact surface as bonded (always): KEYOPT(12) = 5

Note:

- Degrees of freedom (KEYOPT(1))
- Contact algorithm (defaults to augmented Lagrangian) (KEYOPT(2))
- Stress state when superelements are present (KEYOPT(3)) for 2-D surface-to-surface contact. Contact model for node-to-surface contact.
- Location of contact detection point (KEYOPT(4))
- CNOF Automated adjustment (KEYOPT(5))
- Time step control (KEYOPT(7))
- Asymmetric contact selection (KEYOPT(8))
- Effect of initial penetration or gap (KEYOPT(9))
- Contact stiffness update (KEYOPT(10))
- Shell thickness effect (KEYOPT(11))
- Behavior of contact surface (rough, bonded, etc.) (KEYOPT(12))

Rigid Surface Constrained:



Material Properties:

Accurate temperature calculation is critical to the FSW process because the stresses and strains developed in the weld are temperature-dependent. Thermal properties of the 304L steel plates such as thermal conductivity, specific heat, and density are temperature-dependent. Mechanical properties of the plates such as Young's modulus and the coefficient of thermal expansion are considered to be constant due to the limitations of data available in the literature.

It is assumed that the plastic deformation of the material uses the von Mises yield criterion, as well as the associated flow rule and the work-hardening rule. Therefore, a bilinear isotropic hardening model (TB, BISO) is selected.

The following table shows the material properties of the workpiece:

Material Properties of the Plates						
Young's modulus	193 GPa					
Poisson's ratio	0.3					
Coefficient of thermal expansion	18.7 µm/m °C					
Bilinear Isotropic Hardening Constants (TB,BISO)						
Yield stress [1]	290 MPa					
Tangent modulus [1]	2.8 GPa					
Temperature Dependent Material Properties						
Temperature (°C)	0	200	400	600	800	1000
Thermal Conductivity (W/m °C)	16	19	21	24	29	30
Specific Heat (J/Kg °C)	500	540	560	590	600	610
Density (Kg/m^3)	7894	7744	7631	7518	7406	7406

The TBDATA command defines the yield stress and tangent modulus.

The fraction of the plastic work dissipated as heat during FSW is about 80 percent. Therefore, the fraction of plastic work converted to heat (Taylor-Quinney coefficient) is set to 0.8 (MP,QRATE) for the calculation of plastic heat generation in the workpiece material.

To weld a high-temperature material such as 304L stainless steel, a tool composed of hard material is required. Tools made from super-abrasive materials such as PCBN are suitable for such processes, and so a cylindrical PCBN tool is used here.
The following table shows the material properties of the PCBN tool:

Young modulus	680 GPa
Poisson's ratio	0.22
Thermal Conductivity	100 W/m °C
Specific Heat	750 J/Kg °C
Density	4280 Kg/m^3

Boundary Conditions:

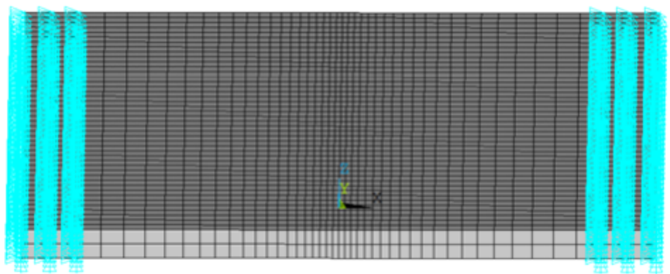
I. **Thermal Boundary Conditions:**

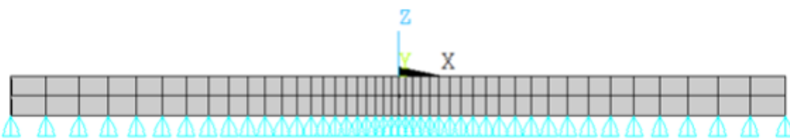
The frictional and plastic heat generated during the FSW process propagates rapidly into remote regions of the plates. On the top and side surfaces of the workpiece, convection and radiation account for heat loss to the ambient. Conduction losses also occur from the bottom surface of the workpiece to the backing plate.

Available data suggest that the value of the convection coefficient lies between 10 and 30 W/m2 °C [1, 2, 3] for the workpiece surfaces, except for the bottom surface. The value of the convection coefficient is 30 W/m2 °C for workpiece and tool. This coefficient affects the output temperature. A lower coefficient increases the output temperature of the model. A high overall heat-transfer coefficient (about 10 times the convective coefficient) of 300 W/m2 °C is assumed for the conductive heat loss through the bottom surface of the workpiece. As a result, the bottom surface of the workpiece is also treated as a convection surface for modeling conduction losses. Because the percentage of heat lost due to radiation is low, radiation heat losses are ignored. An initial temperature of 25 °C is applied on the model. Temperature boundary conditions are not imposed anywhere on the model.

II. **Mechanical Boundary Conditions:**

The workpiece is fixed by clamping each plate [1]. The clamped portions of the plates are constrained in all directions. To simulate support at the bottom of the plates, all bottom nodes of the workpiece are constrained in the perpendicular direction (z direction).





<u>Loading:</u>
The FSW process consists of three primary phases:

1. **Plunge** -- The tool plunges slowly into the workpiece
2. **Dwell** -- Friction between the rotating tool and workpiece generates heat at the initial tool position until the workpiece temperature reaches the value required for the welding.
3. **Traverse (or Traveling)** -- The rotating tool moves along the weld line.

During the traverse phase, the temperature at the weld line region rises, but the maximum temperature values do not surpass the melting temperature of the workpiece material. As the temperature drops, a solid continuous joint appears between the two plates.

For illustrative purposes, each phase of the FSW process is considered a separate load step. A rigid surface constraint is already defined for applying loading on the tool. The following table shows the details for each load step:

Load Step	Time Period (sec)	Loadings on Pilot Node	Boundary Condition
1	1	Displacement boundary condition	UZ = -7.95E-07 m
2	5.5	Rotational boundary condition	ROTZ = 60 RPM
3	22.5	Displacement and rotational boundary conditions together on the pilot node	ROTZ = 60 RPM UY = 60.96E-03 m

The tool plunges into the workpiece at a very shallow depth, then rotates to generate heat. The depth and rotating speeds are the critical parameters for the weld temperatures.. The tool travels from one end of the welding line to the other at a speed of 2.7 mm/s.

<u>Results:</u>