Trailing edge failure analysis of a friction pad in a clutch using thermal fluid structure interaction with LS-DYNA® ICFD solver

Amit Nair[1], Iñaki Çaldichoury[2]

[1]Ansys
[2]Ansys

1 Introduction

A Clutch is a mechanical link used to transmit torque from engine to transmission and typically rotates at very high RPMs. The clutches continuously engage with friction pads to transmit power for motion and only disengage when a gear ratio change is required. During this process of engaging and disengaging the clutch goes from stationary to moving instantaneously. A combination of friction pads and disks are used to transmit the power. There is a significant increase in temperature due to friction between the pads and plate at transition and during rotation. This temperature increase leads to thermal expansion of parts and can cause uneven shape changes. The deformation leads to increase in frictional energy and eventual rise in heat generation. Friction and temperature along with pressure applied during the high rpm rotation leads to high probability of failure at the leading edge of the pads. Uneven distribution of heat can cause failure in the friction pads. To alleviate the effect of temperature, lubricating oil is injected via channels in the friction pad.

To study this complex physics a 2-way coupled FSI solution with LS-DYNA® can provide design guidance in reducing damage and failure. This paper will describe the steps involved in the approach and show the reliability of the approach.

2 Wet clutch model

Clutches in transmission assemblies have a variety of designs. Of the several types available a wet clutch is a design that sits in a pool of oil or has lubrication that flows through tracks in the friction plate. [1]. The friction plate is generally made of composite materials and the tracks can have complex designs to dissipate heat evenly. A pressure pad is pushed to engage the friction pads. Once the pressure is applied the disc engages with the friction pad and starts to rotate at the same speed as the disc. To design the friction pad shape and the flow channels is important to dissipate heat well. Figure 1 shows a typical wet clutch and the lubrication path.

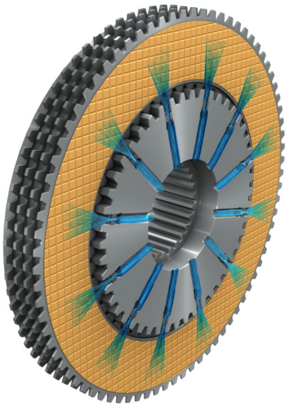


Figure 1: Typical Wet clutch (Image source:https://www.ortlinghaus.com)

3 Solution requirement

As stated in the introduction multiple designs friction plate designs need to be studied to optimize the heat dissipation pattern. Finite element modelling with coupled physics is needed to understand this behaviour. Due to uneven heat dissipation, there is possibility of damage and failure in the friction pads. A capability to predict this damage is also needed. Solid mechanics along with CFD for oil flow and simultaneous Conjugate heat transfer needs to be used. LS-DYNA with its tightly coupled Multiphysics offers a unique solution that can be used for steady state or transient flow with conjugate heat transfer and link both to the structural solver. This solution methodology will help in

1. Analysing the issue and understand the physics
2. Compare designs and optimize channels for oil flow and injection velocity
3. Predict damage and failure of the friction pads

4 Model setup and description

The finite element model is show in Figure 2. The structural model consists of friction pad mounted on a structure. A pressure pad is used to apply the load on the friction pad.

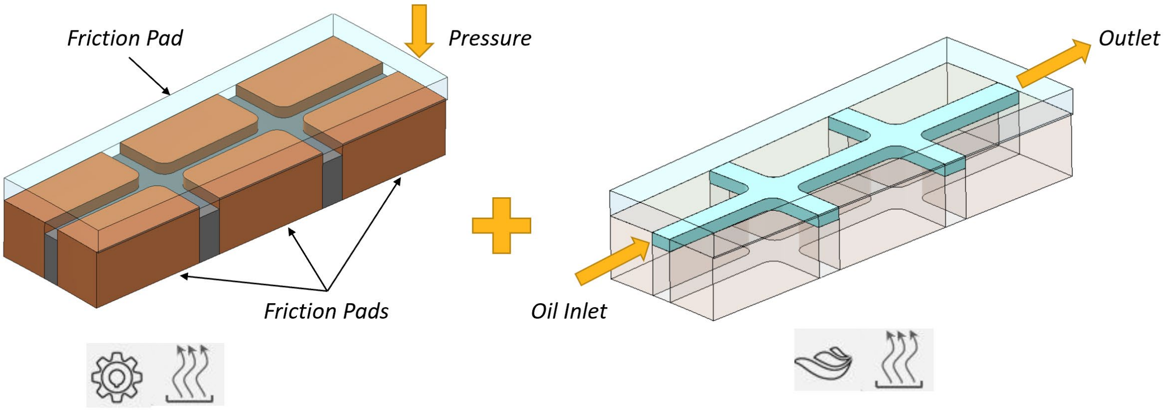


Figure 2: Model Setup

4.1 Structural model and set up

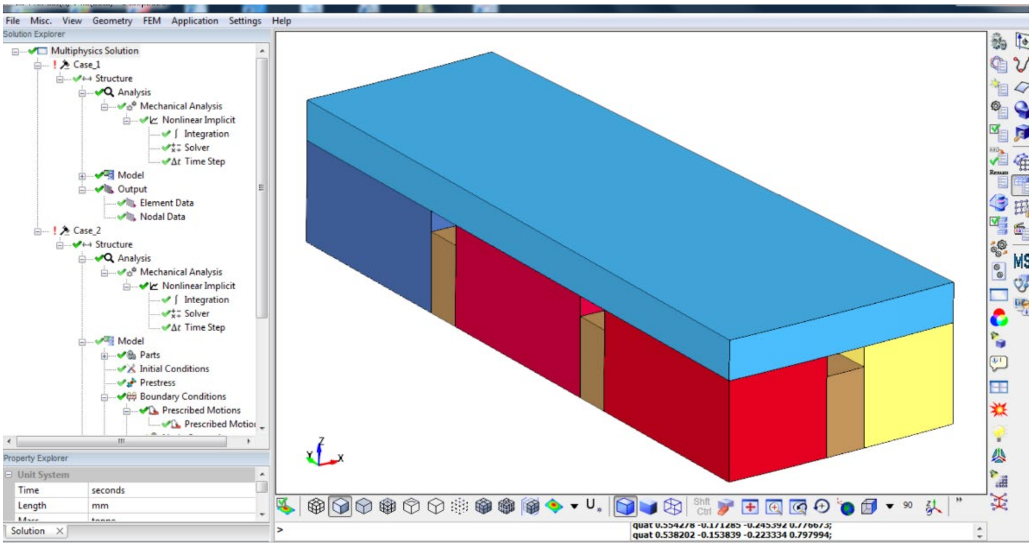
*4.1.1 The model is set up in two stages using *CASE cards. This is done so that we can run the flow field as a steady state flow rather than transient. It aids in reducing complexity and reduce simulation time. On the structural side to further simplify the model we use sectorial symmetry. The bottom of the friction pad is fixed in all 6 degrees of freedom using *BOUNDARY_SPC_SET. The pressure pad is used to engage the friction pad. This is done by applying either a pressure load or a displacement boundary condition (BC). The side faces of the two pads and the plate have symmetric boundary conditions using the same card mentioned above. The displacement BC is applied using *BOUNDARY_PRESCRIBED_MOTION. More details on the simulation are described in section 6.*

*4.1.2 Material model chosen for all the parts is *MAT_PLASTIC_KINEMATIC. Failure and damage modelling can be added based on the material data available. With the simplest form being effective plastic strains to a more complex triaxiality based damage and failure prediction using MAT_ADD_GISSMO can be used. For the purposes of this paper and due to time limits failure modelling was not studied but will eventually be added. Thermal material data is attached to the model using *MAT_THERMAL_ISOTROPIC where we specify the specific heat capacity and thermal conductivity. Thermal expansion coefficient is defined using *MAT_ADD_THERMAL_EXPANSION. This is attached to every material card using its ID. An initial temperature state of 354K is applied to all the nodes using *INITIAL_TEMPERATURE_SET*

*4.1.3 Contact is defined using *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL. This helps in* *specifying heat generation due to frictional effects and to transfer the heat due to conduction. The amount of frictional sliding energy converted to heat can be controlled by a scale factor FTOSLV.*

4.1.4 *The model is set up as implicit transient load case since the mesh size is small and to run explicit the analysis would take longer due to smaller timestep. To set the model as an implicit transient case *CONTROL_IMPLICIT_DYNAMICS is used. The other implicit control cards that are used are *CONTROL_IMPLICIT_GENERAL, *CONTROL_IMPLICIT_AUTO, *CONTROL_IMPLICIT_SOLUTION and *CONTROL_IMPLICIT_SOLVER. The options are mostly standard and can be found in Appendix P of LS-Dyna user manual Volume I.*

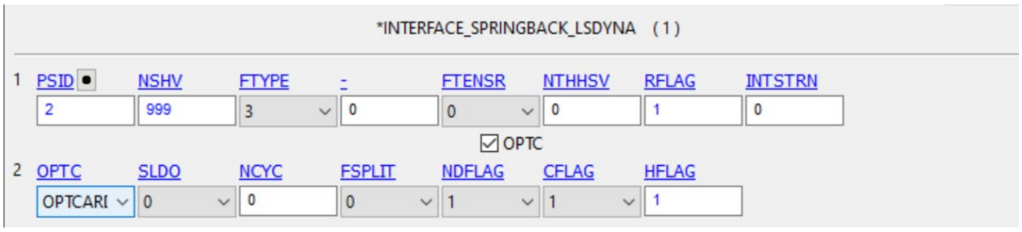
4.1.5 *LS-Prepost is used to set up the model into the stages. Figure 3 shows the solutions explorer available in LS-Prepost. The solutions explorer is accessible from the top menu "view". Mesh is loaded into the module. All model attributes like material, element formulation, contact, controls can be easily defined in the interface. The interface helps a novice user of LS-Dyna to get up to speed with running a simulation. It helps experts in a quick set up of the job, but advanced cards will either need to be entered manually or using the traditional keyword manager available in LS-Prepost. The solutions explorer sets up the cards for stage 2 of the solution too. Stage 2 requires *INTERFACE_SPRINGBACK_LSDYNA to transfer deformation, stress, strain, contact forces and temperature data from Stage 1. The Output feature in LS-Prepost is used to get the temperature at every node at the end of Stage 1 over to the next stage. Figure 4 shows the Interface card that is used.*



Time seconds
Length mm

```
quat 0.538202 -0.153839 -0.223334 0.797994;
```

Figure 3: LS-Prepost Solutions Explorer set up



*INTERFACE_SPRINGBACK_LSDYNA (1)

1	PSID ●	NSHV	FTYPE	-	FTENSR	NTHHSV	RFLAG	INTSTRN
	2	999	3	0	0	0	1	0
					☑ OPTC			
2	OPTC	SLDO	NCYC	FSPLIT	NDFLAG	CFLAG	HFLAG	
	OPTCARI	0	0	0	1	1	1	

Figure 4: Interface springback card

5 The CFD and Conjugate heat transfer analysis (CHT)

For the current application, both the incompressible CFD (ICFD) and the solid mechanics thermal solvers will be coupled together. The ICFD solver starts by solving the Navier Stokes equations using a fractional step approach. Thanks to the incompressibility hypothesis, the heat equation is solved separately. Its expression in the fluid adds a convection term representing the fluid's velocity effects.

When conjugate heat transfer is involved, a single system is assembled with the two domains, fluid and combined, resulting in a monolithic approach. The interface between the fluid and the solid is typically handled by a constraint method with the fluid nodes ‘seeking’ the closest solid face before being projected on it and creating a new constraint condition. This method is considered robust and allows for an accurate representation of an ideal thermal contact between two interfaces.

One of the challenges of conjugate heat transfer simulations is dealing with the numerical cost associated with solving the complete coupled CFD thermal analysis. In a fully transient simulation, the required timestep for the Navier Stokes equations solve is often in the vicinity of 1.e-4 seconds while the total process can take several dozens of seconds or minutes. For this reason, a decoupled approach is adopted. Since for this application the influence of temperature change on the fluid quantities as well as the channel wall deformations can be considered negligible, the fluid velocity and pressure can be solved separately at the beginning of the run. Then, once a steady state has been achieved, the conjugate heat transfer problem can be solved in a transient manner using the results from the steady state analysis to assemble the forced convection term for the heat equation. By proceeding in such a way, calculation times can be greatly improved since after steady state has been reached, only the heat equation remains to be solved. But also, greater timesteps can be used, often in the order of 1.e-2 or higher which further improves solve times.

One further addition in this analysis is the use of thermal periodic boundary conditions. This is a recent addition to the LS-DYNA thermal solver and is again based on a constraint approach. Two sets of segments consisting of the two periodic boundaries need to be defined. Then, on the first segment (usually the one with the finer mesh, if the face meshes are not equal on both sides), a rotation will be applied, and a search of an intersecting faces will be undertaken. An additional temperature equality condition will then be added to the system. It is also worth pointing out that the same keyword (*BOUNDARY_TEMPERATURE_PERIODIC_SET*) can be used to define a sliding thermal contact between two parts, as an alternative to the classic penalty approach used in LS-DYNA.

Finally, for the purposes of this application a new feature has been introduced to the ICFD solver. In traditional FSI applications, a fluid surface mesh is defined as FSI (See *ICFD_BOUNDARY_FSI*). This surface will then seek neighbouring solid faces and its displacements will be permanently bound to the solid displacements. In this application however, the clutch will rotate and slide over the top surfaces of the channel which remain static. The keyword *ICFD_BOUNDARY_FSI_FIXED* therefore needed to be introduced allowing information such as temperature to be exchanged between the solid part and the fluid but forcing the fluid surface mesh to remain static.

5.1 ICFD flow model and set up

Case 2 of the model is the coupled physics with ICFD simulation.

*5.1.1 The steady state flow channel uses the deformed shape of the channels obtained from the previous case. To begin with we have to create the walls of the channel using Shell elements. Element Generate feature is used to create the walls from the faces of the solid elements in the channel. These elements are then converted to *MESH_SURFACE [2] from *ELEMENT_SHELL using FE to MS mesh tool in LS-Prepost. Figure 5 shows a basic difference in the keywords between structure and ICFD. The fluid volume is automatically generated by LS-Dyna. *ICFD_BOUNDARY_FSI_FIXED is used to transfer forces and temperature using 2-way coupling to the structure. As mentioned in the previous section symmetric thermal boundary condition enforces the temperature on both the side faces. Other boundary conditions include:*

- *The fluid inlet velocity is set to 1000 mm/s using *ICFD_BOUNDARY_PRESCRIBED_VEL.*
- *An outlet pressure BC is set to 0 using *ICFD_BOUNDARY_PRESCRIBED_PRE.*
- *Non-slip boundary conditions are specified to the walls using *ICFD_BOUNDARY_NONSLIP.*
- *Initial temperature to the fluid is set to 354K using *ICFD_INITIAL*

5.1.2 The structural BC changes include rotation specified to the pressure pad along with maintaining the pressure on it. The contact definition is the same as discussed in section 4. The deformed shape with the initial stress and strains are included in case2 using case1.lsda file.

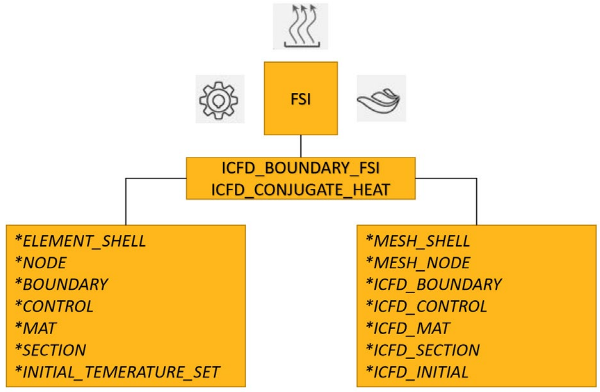


Figure 5: FE and ICFD keywords

6 Results and discussion

6.1 Stage 1

*6.1.1 In stage 1 pressure pad is used to apply load on the friction pads. The case was run using LS-DYNA Implicit Dynamics solver. To improve convergence of implicit calculations a displacement boundary condition (BC) was used instead of a load. A sensor feature is used to track the loads due to contact and the displacement BC is switched off based on a contact load of 3MPa. This is done using *SENSOR_CONTROL, *SENSOR_DEFINE and *SENSOR_SWITCH cards. The pressure pad shown in Figure 6 is nearly twice in size to the plate and the friction pad. This is created to facilitate continuous contact between the pressure pad and the friction pad.*

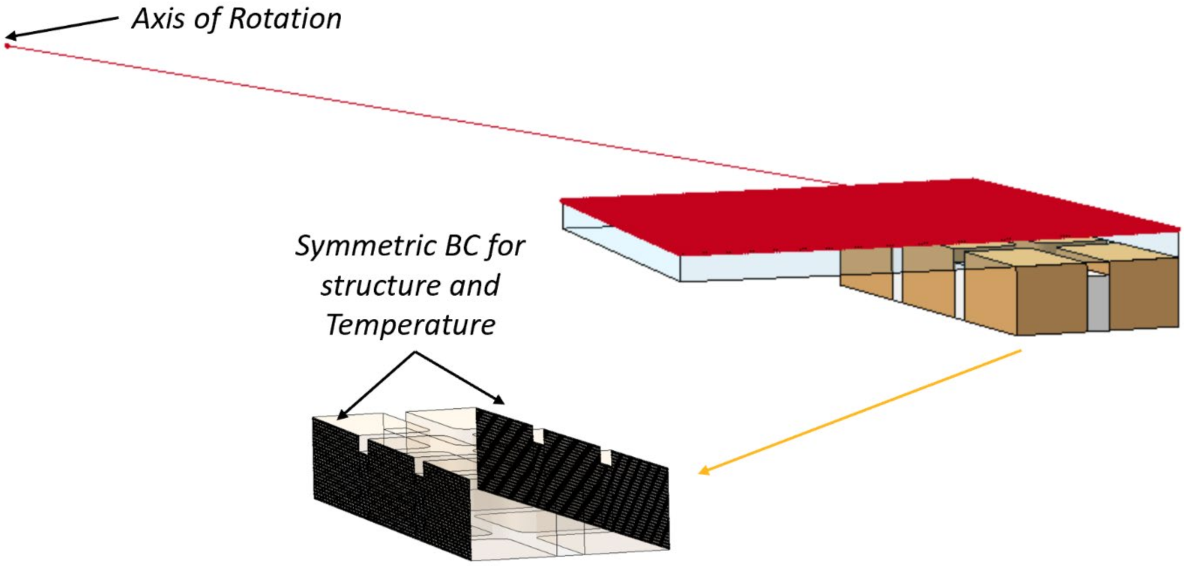
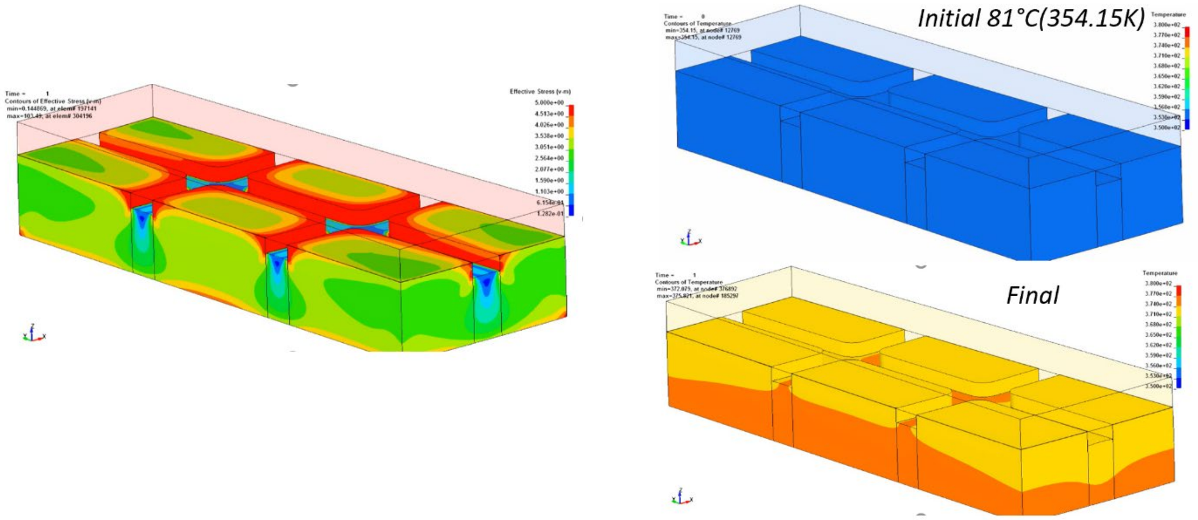


Figure 6: Stage 1 BC Modifications

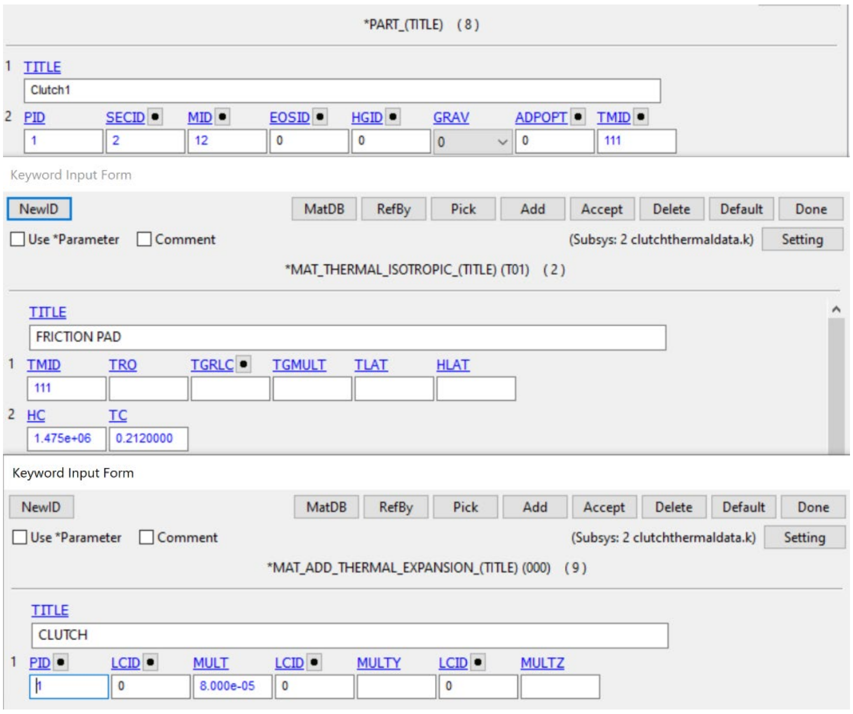
*6.1.2 A *CONSTRAINED_NODAL_RIGID_BODY_SPC (CNRB_SPC) is used to define translational and rotational BCs to specify only Z displacement of the pad. The pressure applied increases the temperature distribution in the parts due to mechanical work done. The thermal expansion of the parts defined in *MAT_ADD_THERMAL_EXPANSION and referred to by *PART ids aids in the increase in the temperature. There is no fluid flow in this stage so there is no cooling. Figures 7 and 8 show the keywords used and the results of stress and temperature distribution* *in this stage. Due to the symmetric structure and thermal BCs the opposite sides of the part as shown in Figure 6 have similar temperature distribution and stress distribution patterns.*



VonMises Stress

Temperature Distribution

Figure 7: Stress and Temperature distribution


*PART_(TITLE) (8)

1	TITLE							
	Clutch1							
2	PID	SECID ●	MID ●	EOSID ●	HGID ●	GRAV	ADPOPT ●	TMID ●
	1	2	12	0	0	0	0	111

(Subsys: 2 clutchthermaldata.k)

*MAT_THERMAL_ISOTROPIC_(TITLE) (T01) (2)

	TITLE					
	FRICTION PAD					
1	TMID	TRO	TGRLC ●	TGMULT	TLAT	HLAT
	111					
2	HC	TC				
	1.475e+06	0.2120000				

(Subsys: 2 clutchthermaldata.k)

*MAT_ADD_THERMAL_EXPANSION_(TITLE) (000) (9)

	TITLE						
	CLUTCH						
1	PID ●	LCID ●	MULT	LCID ●	MULTY	LCID ●	MULTZ
	1	0	8.000e-05	0		0	

Figure 8: Cards for thermal data input

6.2 Stage 2

6.2.1 Deformation input along with contact pressure to stage 2 is included in the set up using case1.lsda. As mentioned in the set up in section 4 a rotational BC is applied to the CNRB_SPC. The larger dimension of the pressure plate helps to maintain contact throughout the simulation. The fluid domain includes the side walls of the friction pad along with inlet and outlet walls as *shown in Figure 9. The fluid surfaces do not change shape. This is an approximation made in assumption that there is no significant deformation in the friction pad side walls. This decision was also based on observing the total deformation of the friction pad was close to 0.2 mm. If there is significant deformation, then a transient flow simulation would need to be done to maintain FSI conditions. This will increase simulation time and complexity. The inlet flow velocity is set to 1000 mm/s and the other conditions as described in section 4.*

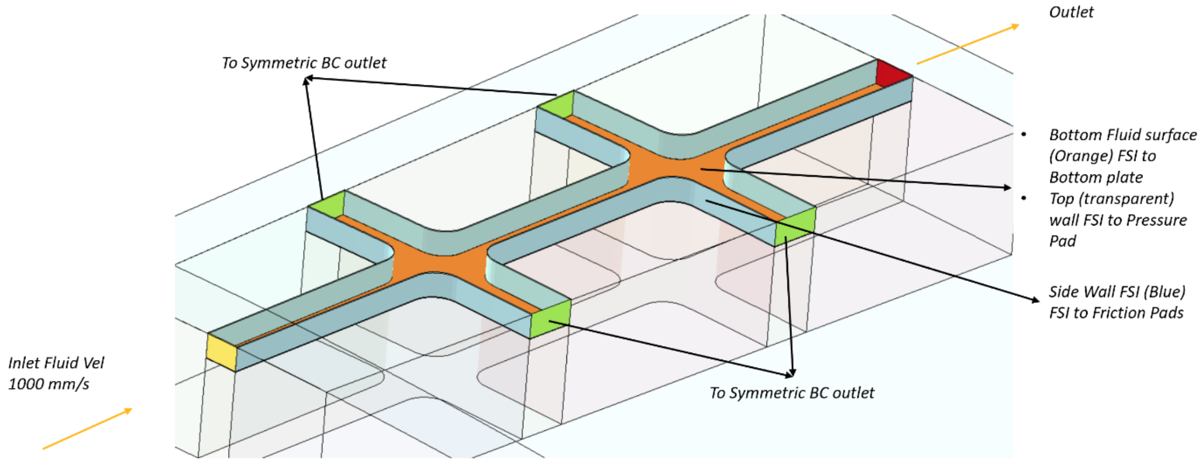


Figure 9: Steady state fluid domain

6.2.2 *Steady state and FSI interactions are discussed in Section 5 above. Steady state is invoked using the card *ICFD_CONTROL_GENERAL (ATYPE=1). Tolerances and iteration limits for steady state conditions are set in *ICFD_CONTROL_STEADY. Monolithic conjugate heat transfer coupling between structure and fluid is invoked using *ICFD_CONTROL_CONJ. To define the interaction between the fluid and structure the card *ICFD_BOUNDARY_CONJ_HEAT is specified for each part. From Figure 9 we can see that the parts added are the side walls along with the top and bottom faces of the fluid. The initial fluid temperature is set to 354.15K using *ICFD_INITIAL. As soon as steady state is reached for the flow the first state of the temperature distribution of the fluid surface shows the interaction between the walls as shown in Figure 10.*

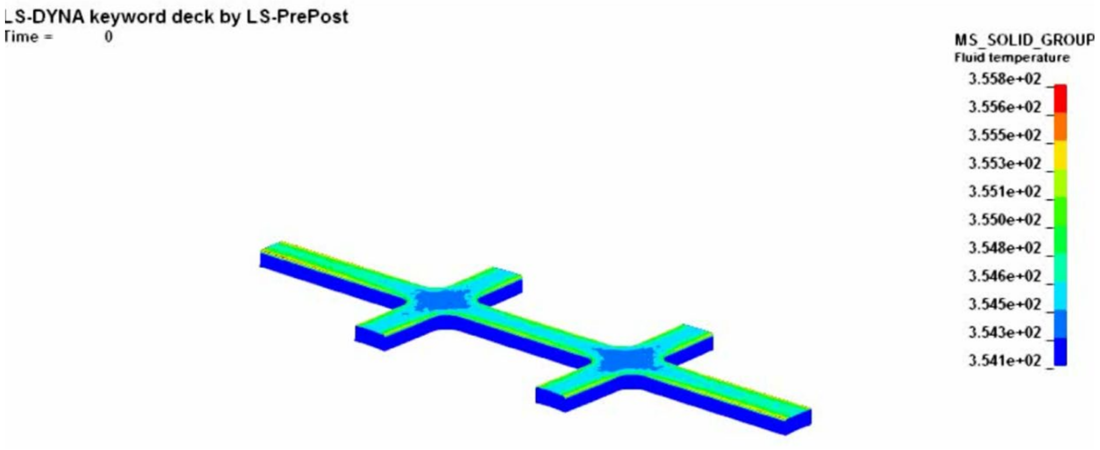


Figure 10: Initial Fluid temperature Distribution

6.2.3 *Temperature distribution of the pressure pad in its initial and final state is shown in Figure 11. As the plate moves over the friction pads and the fluid surfaces it can be seen that the temperature* *change is very evident on the surface. Figure 11 also shows the temperature distribution on the friction plate and the fluid surface.*

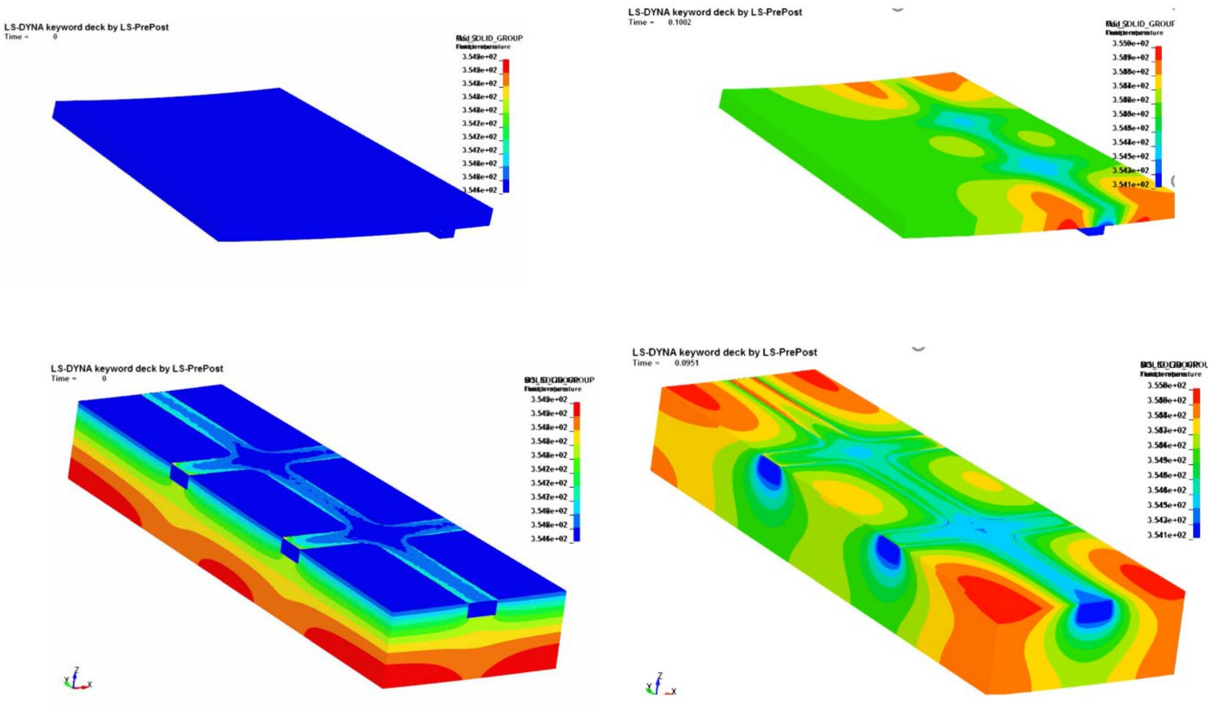
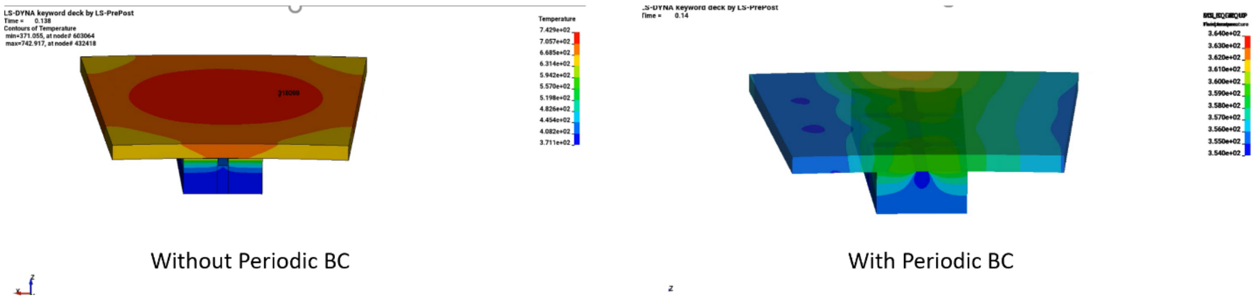


Figure 11: Pressure pad Temperature distribution (Initial and during motion)

6.2.4 *Without the periodic BC for temperature the distribution on the pressure plate is significant was found to be unreasonably high. This difference is evident in Figure 12.*



Without Periodic BC

With Periodic BC

Figure 12: Effect of Periodic Temperature BC

6.2.5 *From the results above it can be seen that the leading edge on the friction pad is experiencing a higher temperature difference compared to the inner side. This is because the fluid cools the inner surfaces of the fluid. This difference in temperature can cause the difference in stress and can lead to degradation of the material and eventual failure of the friction pad surface. The stress distribution is shown in Figure 13.*

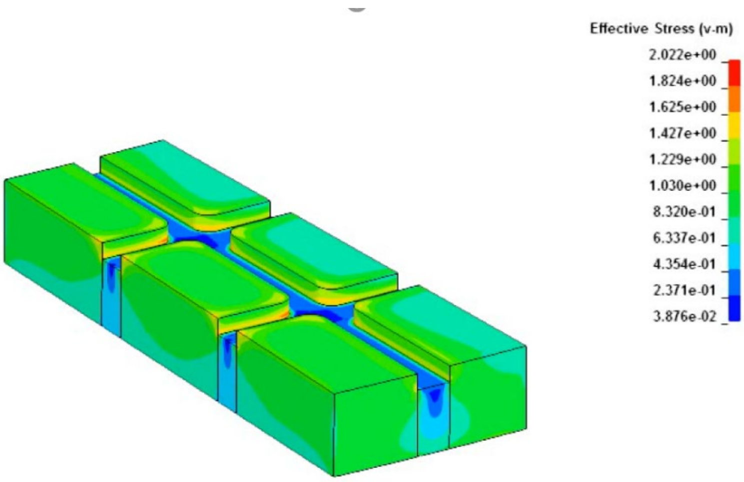


Figure 13: Stress Distribution

7 Summary

A process to study complex multi physics simulation has ben shown in the paper. More work needs to be done to establish the process and study a more detailed model with multiple friction pads. The guidance provided by the simple example shows a lot of promise to adapt the process. The combination of steady state conjugate heat transfer and two-way FSI with deformable parts will help in using the methodology in similar cases.

8 Acknowledgements

Aaron Trisler and Ryan O’Connor of Ansys Corp helped in guidance and reviewing the results. Our sincere gratitude and appreciation go to them for their support.

9 References

[1] Mark Philip Ingram, Thesis presentation on “The mechanisms of wet clutch friction behavior”, Imperial College London, Jan 2010.
[2] LS-Dyna Users manual, Volume III, “Multi-Physics Solvers”, LS-Dyna Dev version.