FlowVision & Abaqus 2-Way Strongly Coupled FSI Simulation of Automobile Tire Aquaplaning

Min-Feng Sung[a]* Chin-Fu Chen [a] Chien-hung Liu[a] Chuan –Jun Yu[b]
[a] *KENDA RUBBER IND. Co. Ltd, Taiwan*
[b] *Samwell Testing Co. Ltd, Taiwan*
*Corresponding Author:bigeyes@kenda.com.tw

Abstact: Aquaplaning of automobile tires simultaneously incorporates various physical phenomena some of which are challenging to model with computer-aided simulation tools. Among those are complex patterned topology and rotational movement of tire, small clearance between ground and tire, free surface flow of water and air and inevitably the elastic nature of tire body and effect of its deformations on fluid flow characteristics.In order to overcome these challenges, Abaqus FE software from SIMULIA and FlowVision CFD package from Capvidia are strongly coupled for bidirectional fluid structure interaction (FSI) co-simulation. This approach is applied to a 205/55/R16 sized tire and same conditions are realized in experimental environment for comparison purposes.Simulation resulthas shown an effect: maximum pressure build-up is at the tire front and once this pressure starts to destabilize by lateral groove; longitudinal groove loses capability of effective draining. Additional observation is that the critical aquaplaning speed of tire increasesin correlation with an increase in the number of lateral grooves.

Keyword: *Tire, FSI, Tire Aquaplane.*

1. Introduction

1.1 Review of Tire Aquaplane researches

Recent developments in high technologies, new materials and advanced manufacturing methods in the motor vehicle industry havetransformed the modern automobile into a universal means oftransport. This popularity has also ledtolotoftraffic accidents, especially in winter. Active safety systems such as Electronic Stability Control (ESC) have considerably increased traffic safety. [1] Strandroth et al. [2] published an article investigating the performance of passenger cars with or without ESC systems in winter. This study examines the effects of studded tires in fatal crashes on roads covered with ice or snow in Sweden. The main study results showed that 64% of fatal crashes on roads covered with ice or snow involved loss-of-control (LOC) as a major component. More than 82% of LOC occurred due to over-steering. Tuononen et al. [3] published a paper that pointed out how to measure the tires contact length on wet-roads. Their results showed the tire contact length on dry roads was very stable, while the tire contact length became very unstable on wet roads. The tire groove function helps toremove water film from the tire contact surface on wet-roads. However, when the water-film thickness becomes higher than groove's depth, hydroplaning effects will occur. In 2013,Guo et al. [4] used Computational Fluid Dynamics (CFD) and Finite Element Method (FEM) techniques to solve the hydroplaning effect. Their result showed that the impact of standing water depth on hydrodynamic pressure is very apparent on a vehicle travelingat90 km/hr.Standingwater at depths ranging from 5 mm to 8 mm presentshydrodynamic pressure increases at the most apparent rate. It was suggested that a car should travel at mid or low speed (65 to 85 km/h) on wet roads. Nakajima et al. [5] used the CFD and FVM techniques to solve hydroplaning effects. They pointed out that simulations could clarify the water flow around the practical tread pattern. The simulation approach allowed developing new tire tread patterns within a short period based on the principle, “make the stream line smooth”. In 2013,Croner et al. [6] published an article indicating that numerical CFD and experiments could demonstrate the wake conditions in a single water slick moving on the road. The simulation results indicated considerable changes in the contact area betweenthe tires and the road surface. Kim et al. [7] recommended using a simplified model to conduct flow field analysis on tire tread patterns in contact with a road surface and then moving away from the road surface. Flow field analysis is then used to analyze the noise.

1.2 The Challenge of numerical solution for Tire Aquaplane

Numerical simulation of water flow around rotated car tireis one of the challenging problems for computational fluid dynamics (CFD). Main difficulties of the problem are:(i)the presence of three phases;liquid, gas, and Flexible bodies, (ii) the tire motion relative to the computation domain, (iii) complex tire shape and large ratio between scales of tirediameter (500-1000 mm) and tread channels (2-10 mm) (iv) hydrodynamic loads resulting in deformation of tire and corresponding variation of tread surface(v) motion of free surface with possible water wavebreaking down.The adequate numerical method allowing to overcome all these difficulties is neededto solve the problem.The described technology is implemented in CFD Package FlowVision [8-10]whichisdeveloped by CAPVIDIA/TESIS. In this paper we present the main features of its application for solutionof car tire aquaplaning.The main object of this paper is therefore to study utilization ofFlowvisionand Abaqus co-simulation technology to analyze tire hydroplaning effects on wet roads.

2. Numerical Tire Modeling and simulation

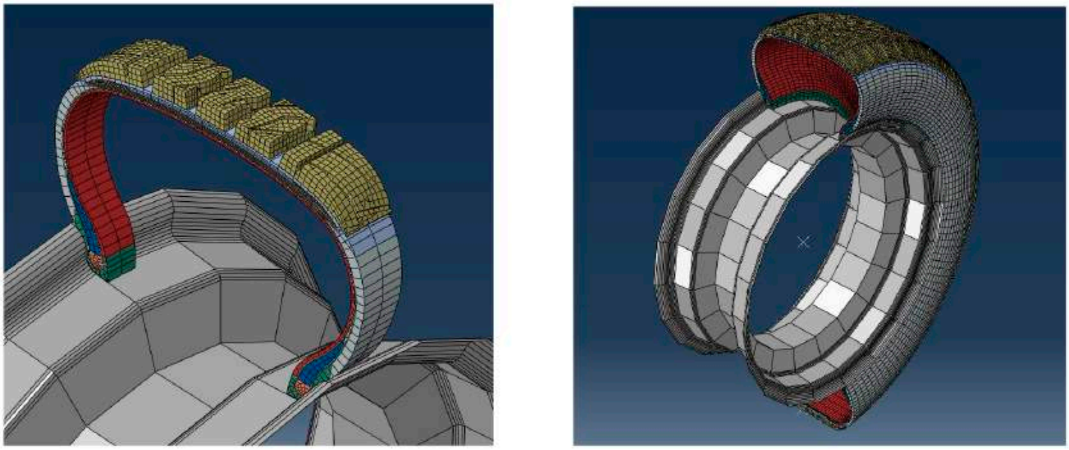
2.1 Characteristics of KEND IDT100 Tire

Many design parameters are critical in producing high performance automobile tires. These parameters include the tire groove pattern, groove depth and width, tread geometric layout, rubber materials, noise level, rolling resistance, performance on both dry and wet roads. Aquaplaning result is the reference base for the same layout design. In addition to above, tire's patternmust bedesignedbyconsideringa variety of conditionssuch as different roadsand driving models. While a specificdesign aiming low resistance increasesfuel efficiency, another design is for better handling and safety on wet roads.

In this paper, KENDA IDT100 has been selected for research. This tire has four continuousditchesto increase driving stabilityand uses OFF-shoulder to reduce the rolling noise. The special design for IDT100 alsohas large block pattern to increase handling performance and reduce the hydroplaning effects.

2.2 Tire FEM Modelingin Simulia Abaqus

The need for reduced computational efforts brings therequirement forthe analysts to simulate only the key features describing the phenomenon under investigation. Material stiffness non-linearity has to be taken into account both for rubber and reinforcements. We can basically split tire intoan axisymmetric structural part and a non-symmetrical part: the tread. Tread geometry is obtained assembling a sequence of pitches (Fig.1).



(a) Tire sector model **(b) Full-Tire model**

Fig.1Tread geometry construction in FEM modeling

Treaded tire model will be subjected to service conditionsusing Abaqus/Standard: once the geometry has been generated taking advantage of periodicity, the tire will be mounted on the proper rim and inflated at a given pressure.

A vertical load is then applied and, using Steady StateTransport analysis, the tire will reach a stationary rolling condition at 65 km/h travelling speed. After then thetire is finally imported into Abaqus/Explicit to continuously be accelerated from 65km/h to 130km/h during certain time interval. In the meantime, the explicit analysis will start co-simulation with FlowVision CFD packagesoftware, to simulate the tire rolling over different thickness of water filmsand investigate the performance of tire wet-grip capabilities (Fig. 2).

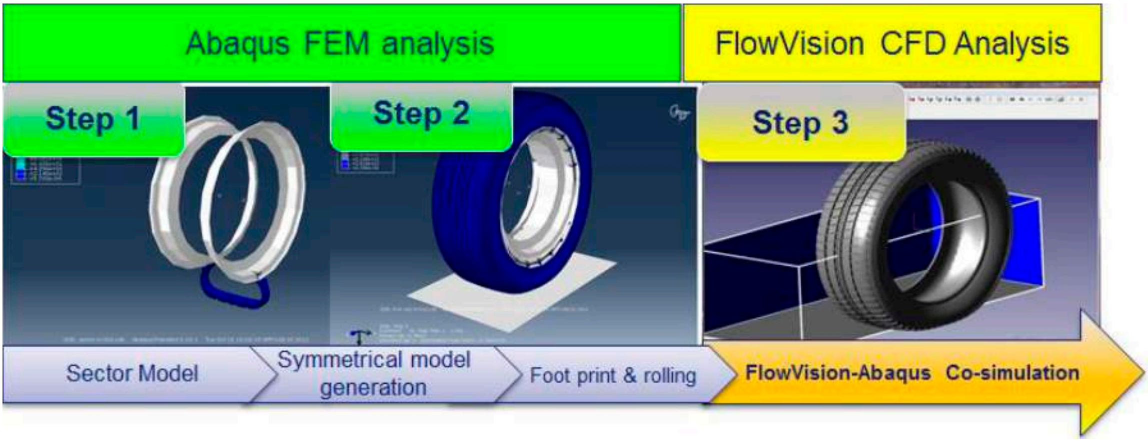


Fig.2 AnalysisProcedure of FlowVision-Abaqus co-simulation for TireAquaplane

2.2 Tire CFD Modelingin CAPVIDIA FlowVision

After the Tire FEM modeling is complete, it can be directly imported into Flowvision.InteractiveFlowVisioninterfaceallows userto create computational domain for tire aquaplane simulation, and specify appropriate boundary conditionsfor the Abaqus -Flowvision 2-way strongly coupledFluid-Structure interaction simulation oftire aquaplane(Fig.3).

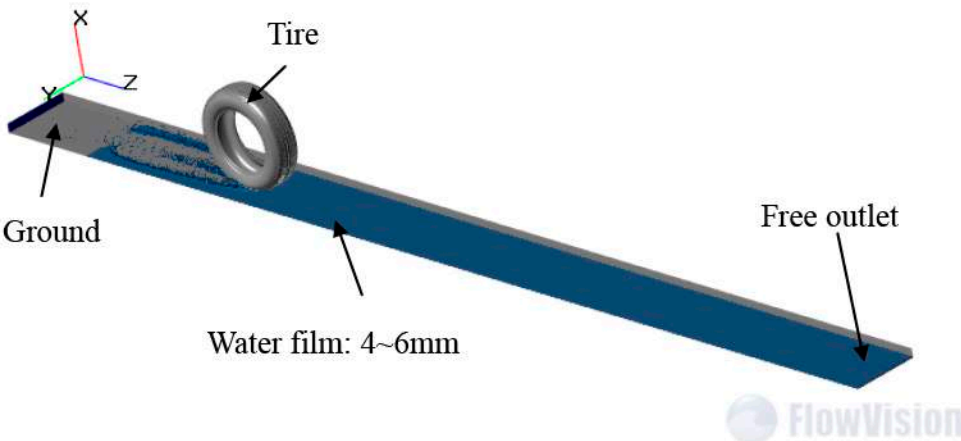


Fig. 3 Computation domain and boundary conditions

The CFD mesh in FlowVision is built fully automatically using the Sub-grid Geometry Resolution method.Those technologiesarevery usefuland convenientfor modeling in presence of air/water free surface flowaround rolling tires which contain very complexgeometries (Fig. 4).

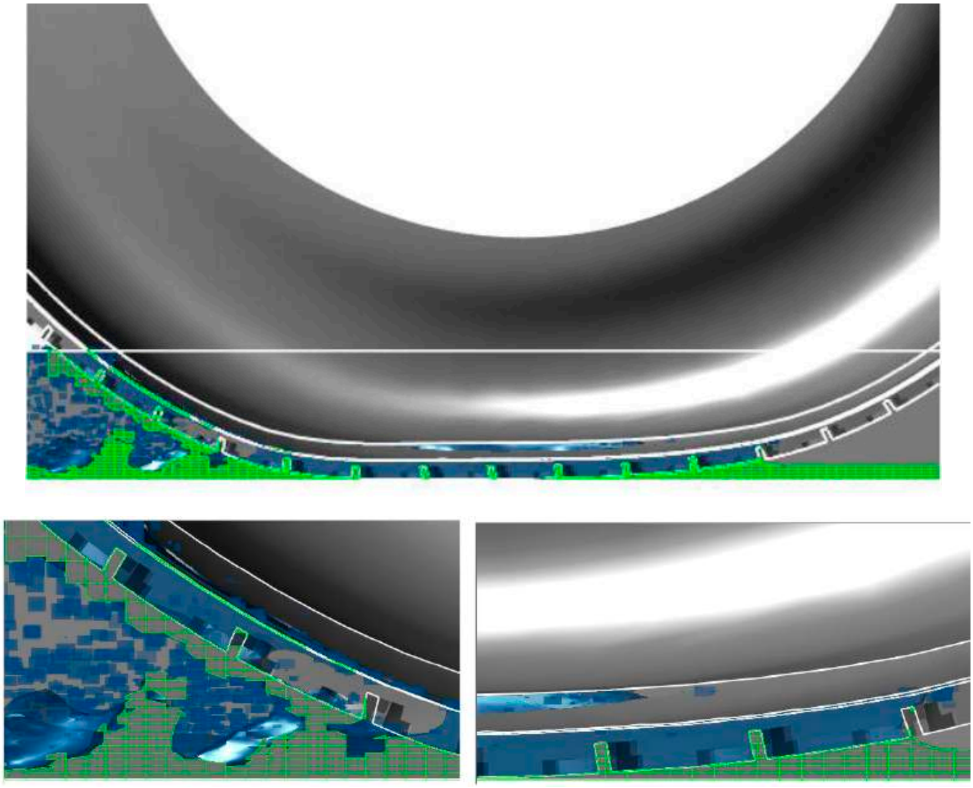


Fig.4Mesh refinementnearcontact areaof tire thread and road

To take into account peculiarities of tire structures shape andmotion of themoving parts, FlowVison providesvery powerful mesh sizingand distributioncontrol tools.Built-ingrid generatorsallows easily resolve the interest area of tire in CFD domain.Generally, the CFD domainin FlowVisionisinitially meshedby uniform or non-uniform schemesas user preferred.Followingthat,localgridadaptation techniquesare utilized to refinethe mesh near the boundary of geometries or the space of geometries. Tire shape to be resolved significantlyincreasesthe requirednumber of cells.The CFD mesh convergence analysis will be discussed in the following chapters.

2.3 FSI Co-Simulation model Setup

To set up a FSI co-simulation is very easy for users who are familiar with Abaqus and FlowVisionas shown in Fig5.User creates Abaqus project for simulating explicit analysis for tire dynamic rolling, In the Abaqusinput file,just the outside surface of the tire body and tire threadsneedsto be defined.This Abaqus input file is imported into FlowVision project to definenecessary boundaryconditions and models forthe CFD Analysis, for example,VOF model for air/water flow. Thelink between FE and CFD meshes is built automatically via Flowvision SGGR Technique.Finally,In the MPManager user defines only the path to Abaqus and FlowVision codes and defines the FSI time step.

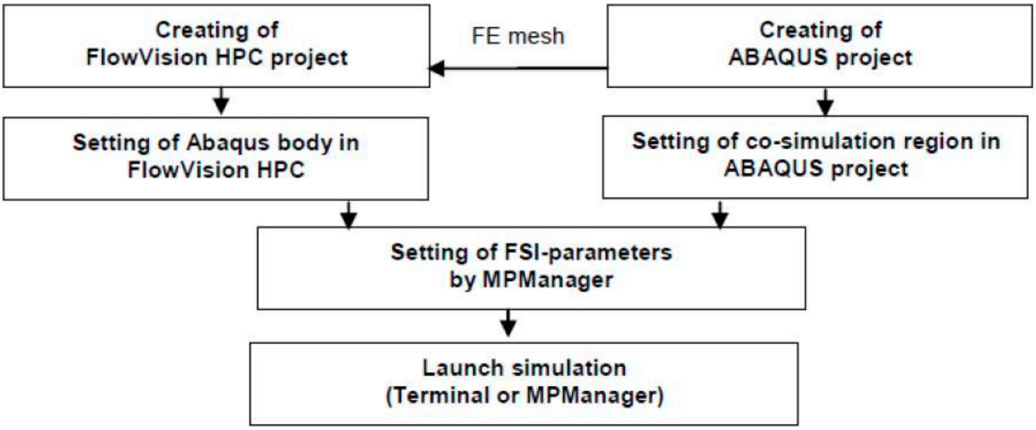


Fig. 5 Co-Simulation DCI arrangement

2.4 Two-Way Strongly Coupled Fluid-Structure interactions

FlowVision combined with SIMULIA Abaqus offer very unique co-simulation capabilities for solving stronglycoupled fluid-structure interaction problems.

The SGGR based grid generation in FlowVision provides natural link between the Fluid and FEA domains. The “Wetted interface” is automatically established and rebuiltdynamically even if the tire is rolling associated with very complex behavior between tire threads and road;tire blockslip and stick-slip.

FlowVision MPM (Multi Physics Manager) provides co-simulation mechanisms addressing most complex FSI simulations. Without any need for a 3rd party component (such as MPCCI), FlowVision MPM drives the process by controlling the data transfer, by starting/restarting both solvers and providing feedback on the simulation progressDirect coupling between Abaqus FEA and FlowVision-HPC CFD code is used. Details of this coupling method is described in FlowVision User’sguide [10]

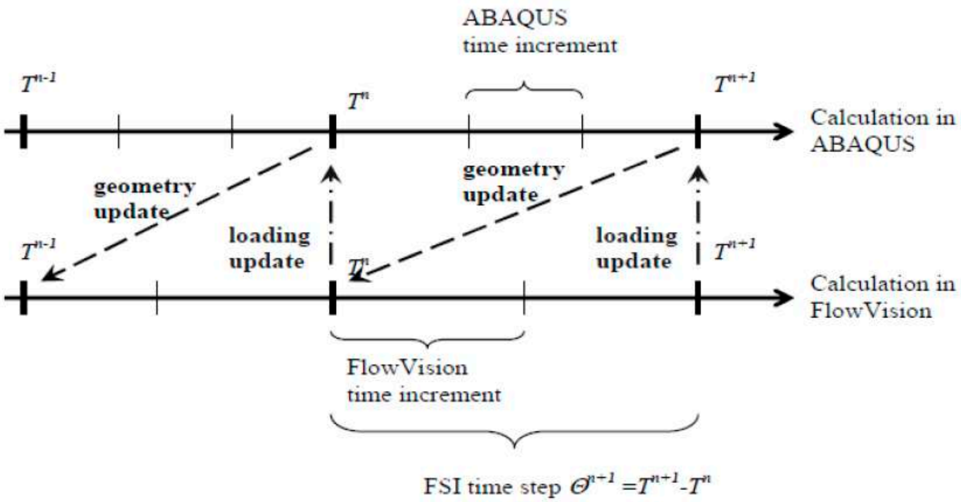
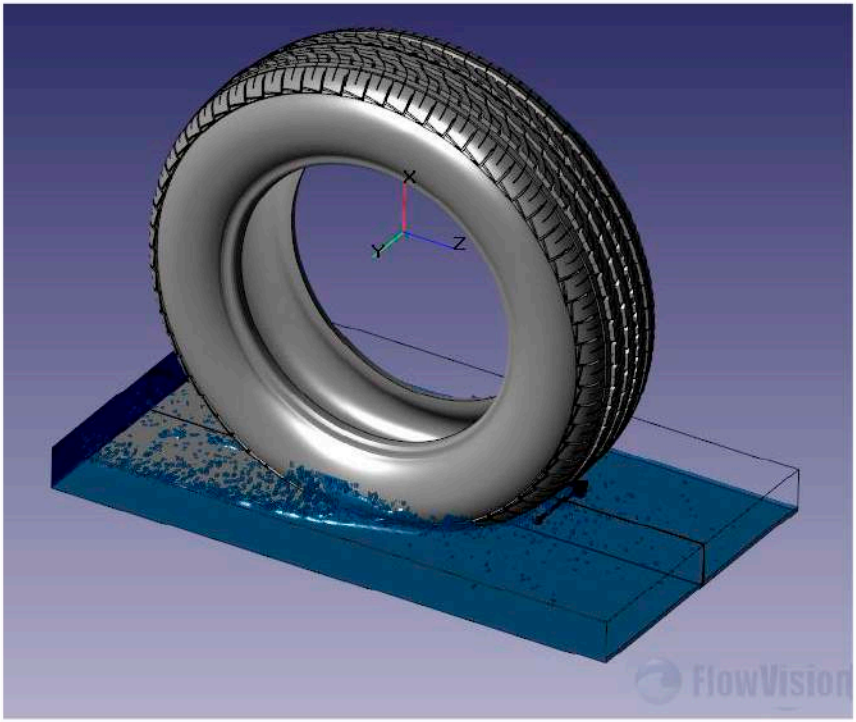


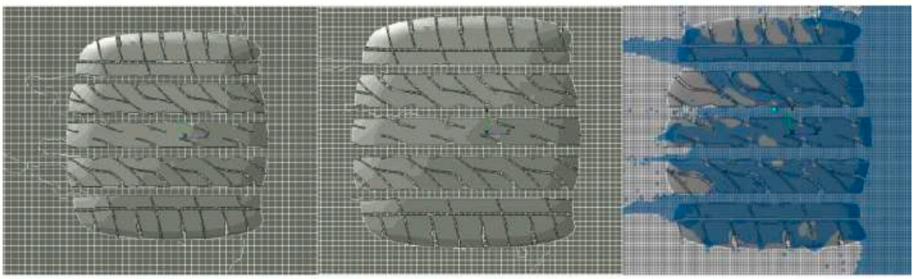
Fig.6 FSI time step for FlowVision and Abaqus

Explicit coupling procedureshown in Fig.6is applied for the time stepping algorithm. The exchanged data is consisting of forces computed in the nodes of the FE mesh, which comes from FlowVision and the node displacement exchanged data coming from Abaqus. The exchange time period (FSI time step) is based on the value of the CFD explicit time step (min h/V over all CFD domainswhere h: cell size, V: fluid speed and Moving boundary speed). As practice shows, for the fast simulation processes, like the analyzed Tire Aquaplaning, the best choice of FSI time step is 1 or 2 times of the CFD explicit time steps(e.g. the scale of time step is about 2.5~3.5e-5s). The FSI time step is started by Abaqus and uses loadings from the previous FSI time step, as shown in Fig.6. FlowVision receives new node coordinates and calculates new loadings.

2.5 CFD computational gridconvergence investigation

An investigation of computational grid convergence is performed to be sure that numerical analysis results in CFD do not depend on mesh quality.During this investigation;several analyses were run with different computational grids for tire at constant traveling speed (Fig.7).





(a) (b) (C)

Fig.7 There different CFD computational gridsnear foot print area:

(a) 0.27 million(b)0.8million(c)4.4 millions

Dependence of lift forceon computational grid is represented in Fig. 8. Lift Forcechanges are not significant when number of mesh reached to0.27 million.

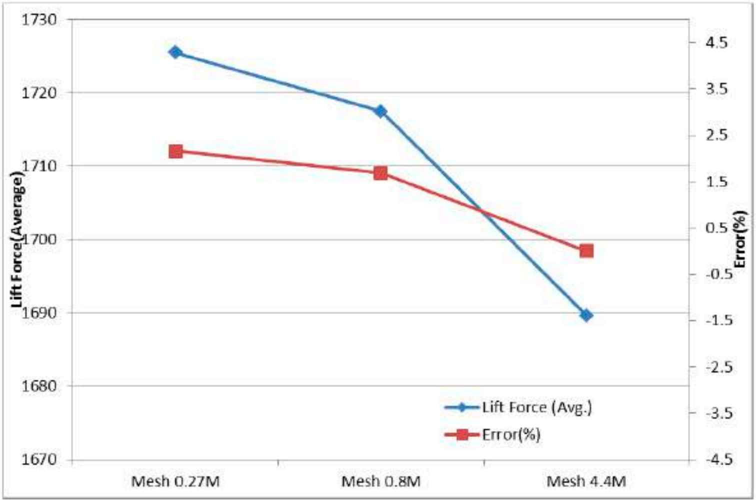


Fig.8 CFD mesh convergence analysis result

It is apparent that as the CFD mesh is refined beyond a cell count of 0.8million cells, the influence of mesh refinement upon the predictions becomeless important. All results given within this paper use a grid with0.8millioncomputational grids.

2.6 Governing equations solved

2.6.1 Fluid dynamics of water

In this study, water is considered as main working fluid and governed by system of equations for incompressible fluids including continuity and Navier-Stokes equations, expressed as following equations (1) and (2):

(1) $$\frac{\partial_p}{\partial_t}+\nabla\cdot(\rho V)=0$$

(2) $$\frac{\partial_p V}{\partial_t}+\nabla\cdot(\rho V\otimes V)=-\nabla P+\nabla\cdot\hat{\tau}_{eff}+\rho F$$

Equations (1) and (2) were used to solve the velocity and pressure of fluid with a decouple process called as *pressure-velocity split process* [8-10]. Here, $\hat{\tau}_{eff}$ is the effective shear stress tensorcaused by viscosity of fluid. The turbulence of water flow around the rolling tire is also consideredin FlowVision.The standard K-ε model is used for some caseswhere equations(3) and (4) areused to solve the turbulent energy and dissipation of fluid.

(3) $$\frac{\partial(pk)}{\partial t}+\nabla\cdot(\rho Vk)=\nabla\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right)+\mu_t\left(G+\frac{\beta}{\mathrm{Pr}_t}g\cdot\nabla T\right)-\rho\varepsilon$$

(4) $$\frac{\partial(p\varepsilon)}{\partial t}+\nabla\cdot(\rho V\varepsilon)=\nabla\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right)+C_1\frac{\varepsilon}{k}\mu_t\left(G+\frac{\beta}{\mathrm{Pr}_t}g\cdot\nabla T\right)-C_2\rho\frac{\varepsilon^2}{k}$$

The model parameters and the expression for generating term G can be rewritten as (5), (6), and (7):

(5)
$$G = D_{ij} \frac{\partial V_i}{\partial_{xj}}$$

(6)
$$D_{ij} = S_{ij} - \frac{2}{3}\left(\nabla \cdot V + \frac{\rho k}{\mu_t} \right) \delta_{ij}$$

(7)
$$S_{ij} = \frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i}$$

The object of this study is to investigate the hydrodynamic force ona rolling tire caused by fluid-dynamics.The units and numerical settings are listed in Table 2. Equation (8) was used to solve the total pressure of fluid, and the total hydrodynamic force onarolling tire is calculated by Equation (9).

(8)
$$P_{tot} = P + P_{hst} + \frac{1}{2}\rho |V_{abs}|^2$$

(9)
$$F_{fluid} = \oint_s (P + P_{hst}) n dS - \oint_s (\mu + \mu_t) \frac{\partial V}{\partial n} dS$$

2.6.2 Free surface flow-Advanced VOFmodel

The free surfaceflowis modeled by the“Advanced VOF model “in FlowVision.Transfer of the water phase boundary is described by the equation for the volume fraction of this (continuous) phase in a computational cell ('Volume ofFluid' = VOF). Variable VOF takes values from 0 (Gas) to 1 (Liquid).A cell, where 0 < VOF < 1, contains a contact surface. In solver, this surface is represented by a set of polygons. Propagation of the volume fraction of water is solved by the transport equation (10).

(10)
$$\frac{\partial F}{\partial T} + V \cdot \nabla F = 0$$

2.7 Computation resource and performance

2.7.1 Domaindecomposition of CFD mesh for parallel computing

In many important computational mechanics applications, the computation adapts dynamically during simulation. Examples include adaptive mesh refinement, transient fluid dynamics, and free surface of multiphase flow calculations. When running these kinds of simulations on a parallel computer, the work must be assigned to processors in a dynamic fashion to keep the computational load balanced.

Fig.9 showsCFD mesh splitting and distribution over 6 processors. When FSI co-simulation is started; FlowVision initially builds computational grid and splitsit into several sub-domains automatically. After then, FlowVision redistributes those domains of computational grid between processors. As seen, some domain (e.g. color in red) is slightly irregularin distribution when compared too thers initially. It is possible to result inunbalanced computational loadsand leads to slowed down calculations. While the dynamic balancing technology is performed, the domain decomposition, for example, the red domain, is now distributed almost equally to others. Dynamic balancing is helpful to balance the loading on each processor during parallel computing,making the calculation more efficient and faster eventually.

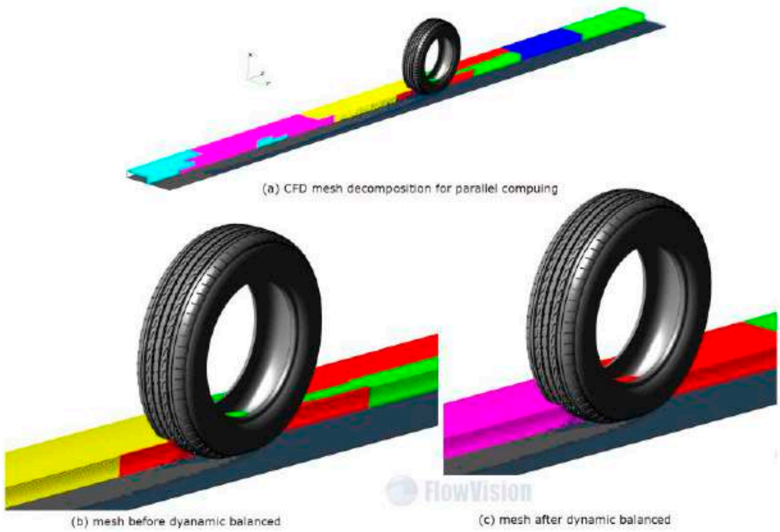


Fig.9 dynamic CFD mesh balancing for parallel computing

2.7.2 Performanceof FSI co-simulation

Fig 10.ShowstheCPU time it takes to iteratefor each stepin Flowvision. As seen, the CPU time is growingwhen the number of step is increased. It is because the flow is relativelysimple atthe beginning, but when the tire cutsthe water, wateris splashed. It makesthe watertoflow with very complicatedform. Therefore, the solver needs more time to reach the designatedresidual(e.g. <1e-5)for CFD simulation.

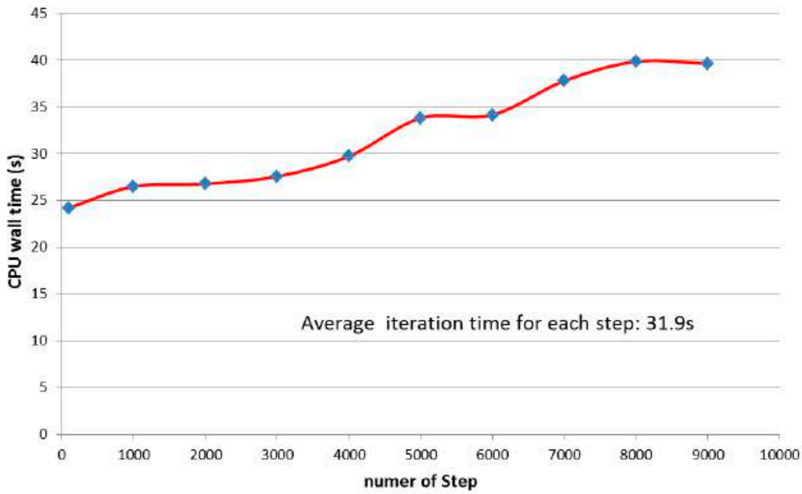


Fig.10 CPU time it takes iteratingfor each stepin FlowVision

Table 1-2 shows the computationaltimeand specification for each task. As seen, three different computers have been used in this study respectively.Owing to the initial grid distribution for each task is same;as a result, the total number of CFD mesh isreasonablyincreased as the water filmbecomes thick.The number of FEA elementsin Abaqus is about 0.23millionand the CFDmeshin FlowVisionis about 0.43 ~0.8 million.

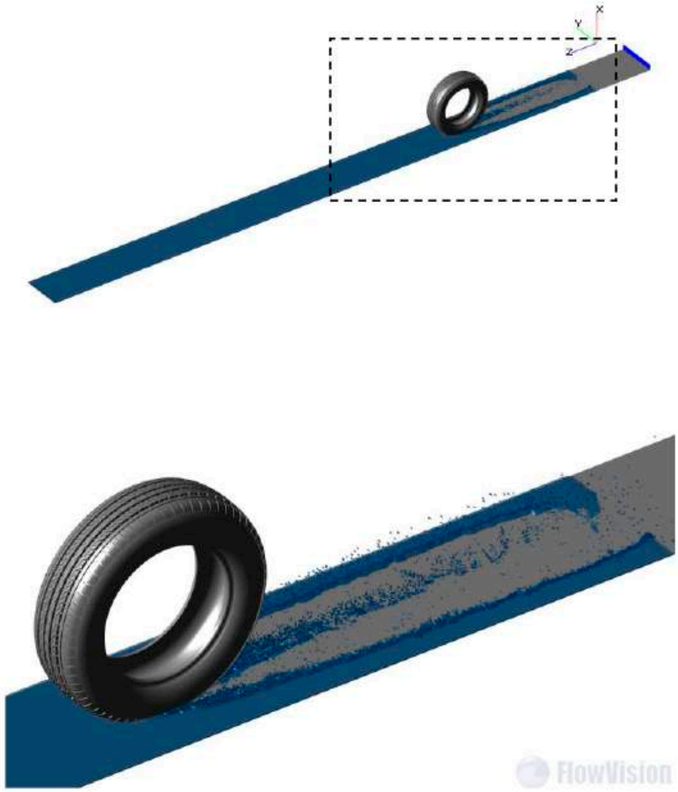
The tire aquaplaning processes was simulated from 0~0.2sec. The time increment for the Abaqus explicit analysis was about 5.46e-8s which correspondingly requires 950,000 increments for reaching the designated time. For the CFD Part, time step in FlowVison was ranging between2.5e-5to 3.5e-5which is automatically calculated by Courant Freidrichs-Lewy (CFL) method via FlowVison. The corresponding requires number of steps to reach the designated time is about 9,000 stepsin CFD.As a result,the total computational time for CFD part is about 80~100 and50~80 hoursfor FEA part.Finally, the total computationa ltime for a complete analysis of FSI co-simulation is about 130 ~190 hr.

Table 1-2 FSI computational timeand specificationfor each task

Water film	CFD mesh	FEA element	Computer Spec.	Flowvision CPU time	Abaqus CPU time
4	430,000	230466	Core i7 3930K 6C/12T	81.9 hr.	62hr.
6	620,000	230466	Xeon E5 2650 V3	81.9hr.	51hr
8	800,000	230466	Core i7 5820K 6C/12T	99.7 hr.	86hr.

3. Simulation Results

Fig.11shows the result of Flowvision-Abaqus co-simulation for Tire Aquaplaning phenomena; the water is drained to the perimeter through two circumferential grooves and then lateral grooves, and the bow wave is generated at the leading edge.



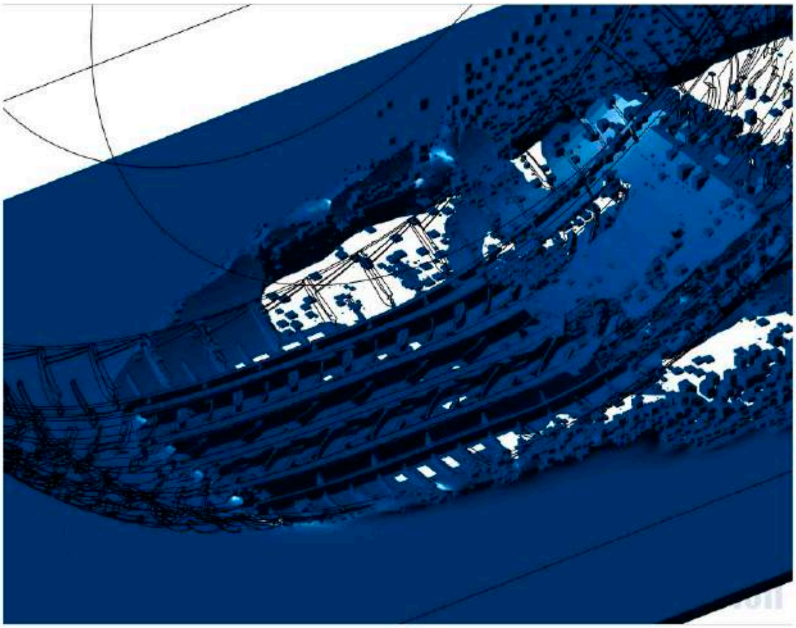


Fig.11 Tire-Aquaplaning calculated by Flowvision-Abaqus co-simulation

3.1 VELOCITY DEPENDENCE OF HYDROPLANING

The time histories of contact forces areshown in Fig.12. Vertical axis in this figure is the contact force between the tire and the road, and the horizontal axis is the speed of the vehicles in the simulation.

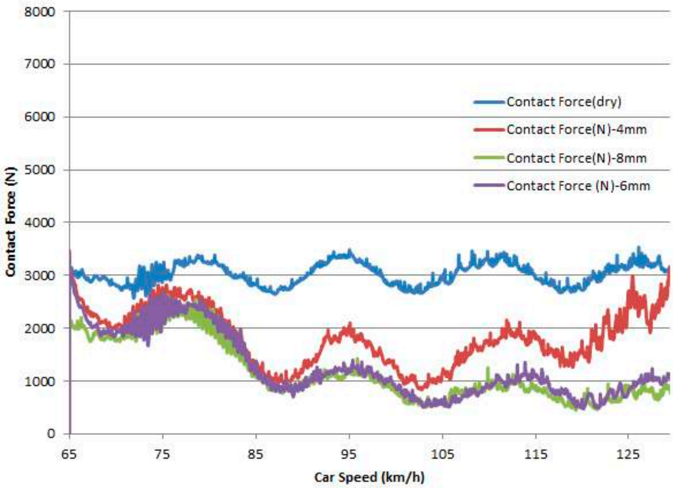


Fig.12 Car speed dependence of contact force between tire thread and road

Since hydroplaning occurs when the hydrodynamic pressure exceeds the contact pressure, the hydrodynamic force is considered as a good criterion for hydroplaning. In order to investigate how the tire is influenced by the wet condition of ground, we calculated four cases which were the rolling tires on dry pavement and the tire rolling on the water with 4, 6and 8 mm thicknesses.

The relationship between vehicle velocity and slip ratio of a tire is shown in Fig.13.The slip ratio of the tire increases gradually as the vehicle velocity, driving in water pool, increases.When vehicle velocity exceeds a critical velocity, slip ratio of tire rises drastically by the onset of hydroplaning.

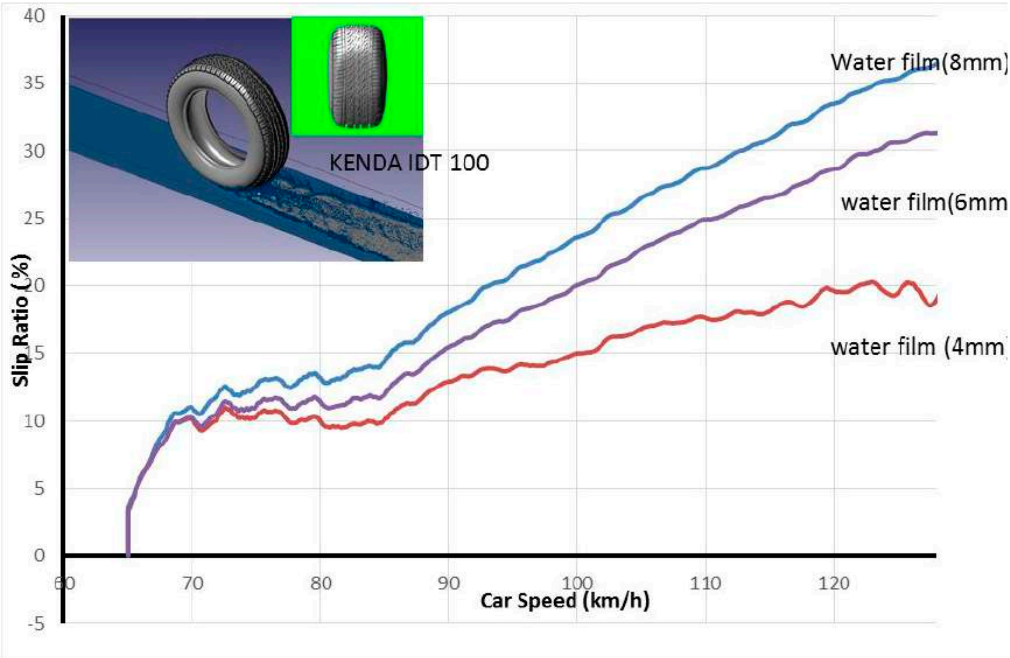
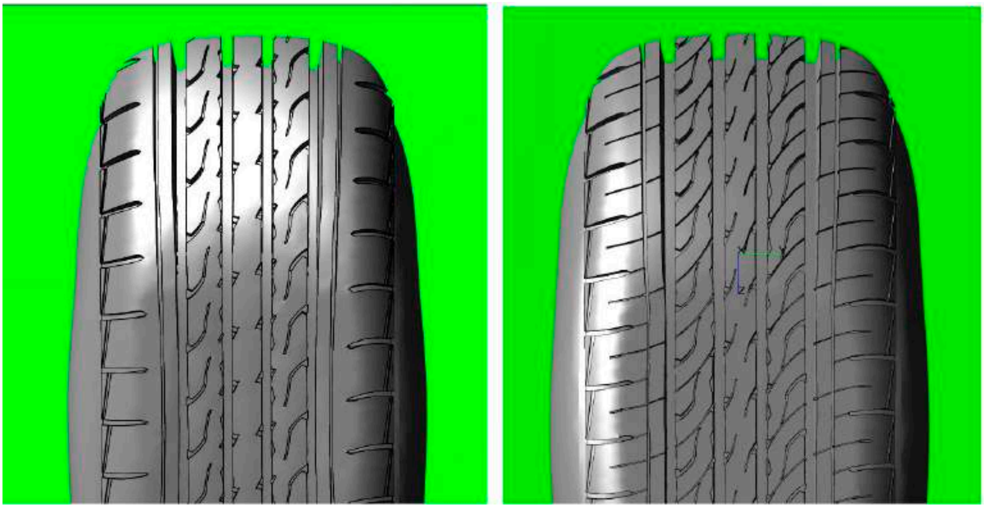


Fig.13 Velocity dependence of Slip Ratio of Tire

3.2 EFFECT OF TREAD PATTERN ON HYDROPLANING

The effect of the tread pattern on hydroplaning has been predicted in this study. The slip ratios of tire with and without lateral grooves for various car speeds are shown in Figure 14-15. As seen, the higher number of lateral grooves the higher critical tire aquaplaning speed can be reached, which is corresponding to conventional knowledge. This is also explaining why the pattern geometry is very important for hydroplaning performance.



(a) Without lateral grooves (b)Withlateral grooves

Fig.14 KENDA IDT100 tire with/ without lateral grooves

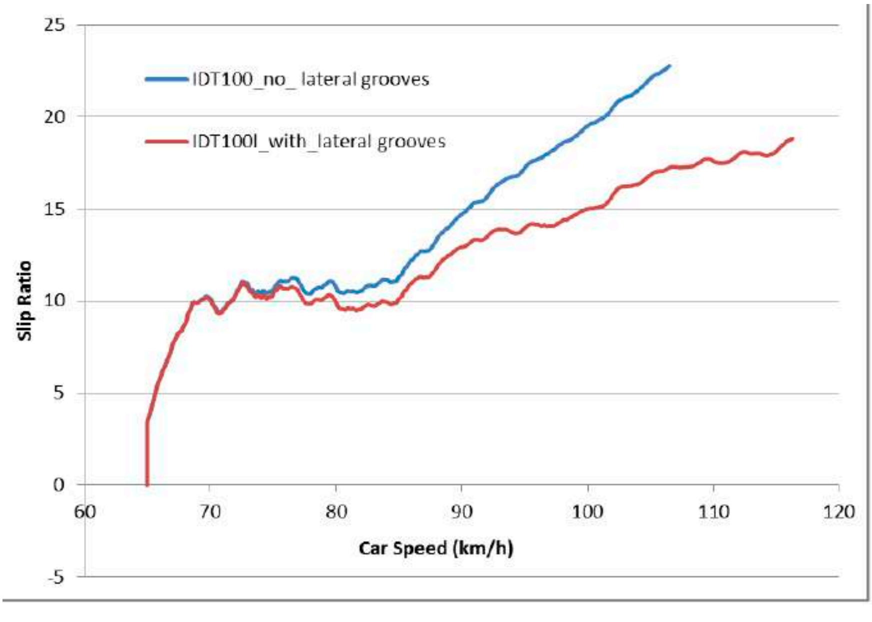
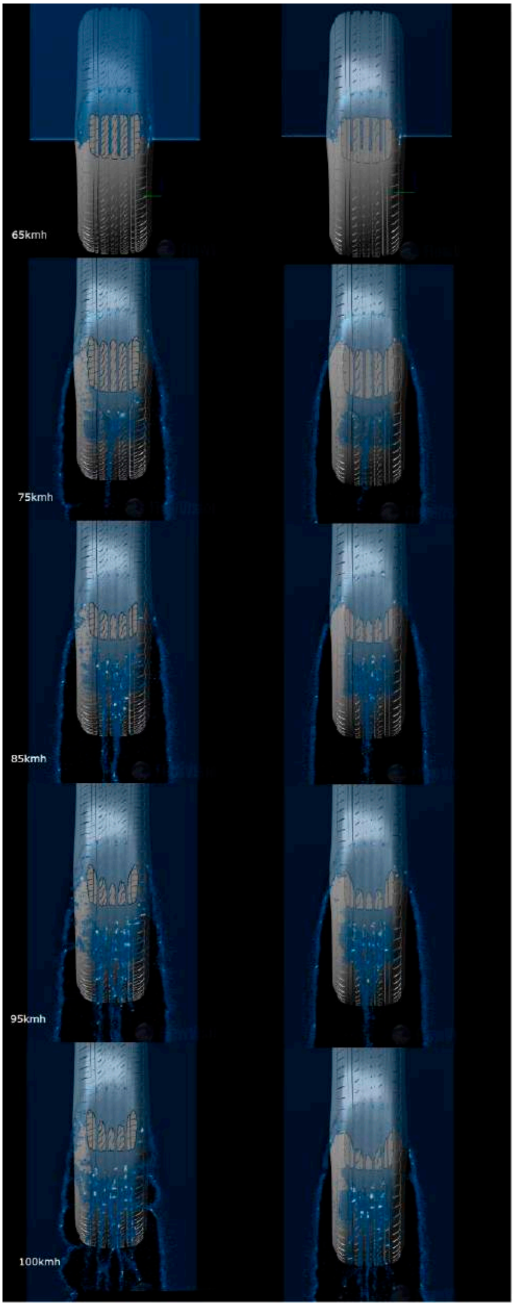


Fig.15 Effect of tread pattern on aquaplaning

The water is drainedby two treadpatternsof various car speeds are shown in Fig. 16. As seen, using the proposed innovate method, FE-CFD direct coupled, for simulation of fluid-structure interaction tire aquaplaning, the result shows very realistic fluid-dynamics of water flow around the tire.

The water streamlinearoundthecontact patch of various car speedsareshownin Fig.17. As seen, the water is drained to thecircumferential direction at the center of the contact patchand to the lateral or slantwise direction at the shoulder ofthe contact patch respectively. Investigatingdetailsofwater flow, onecan recognize that the drained waterflows into the circumferential grooves at the center, andflows into the lateral grooves at the shoulder.

Hydroplaning occurs when the hydrodynamicpressure equals the contact pressure that is almost proportional to the inflationpressure. The comparison ofthe water flow around the contact patch fordifferentvelocitiesis shown in Figure 18. As seen, the hydrodynamic pressure around the leading edge increases with the velocity.In addition, it is obvious that hydrodynamicpressure for the tire with lateral grooves isrelativelysmaller than the tire withoutlateral grooves. It can explain why tire with lateral grooves has better wet-grip performance.



(a) With lateral grooves (b) without lateral grooves

Fig. 16 Waterdrained byIDT 100tireswith/without lateral grooves at different traveling speed (post-processing by FlowVision CFD)

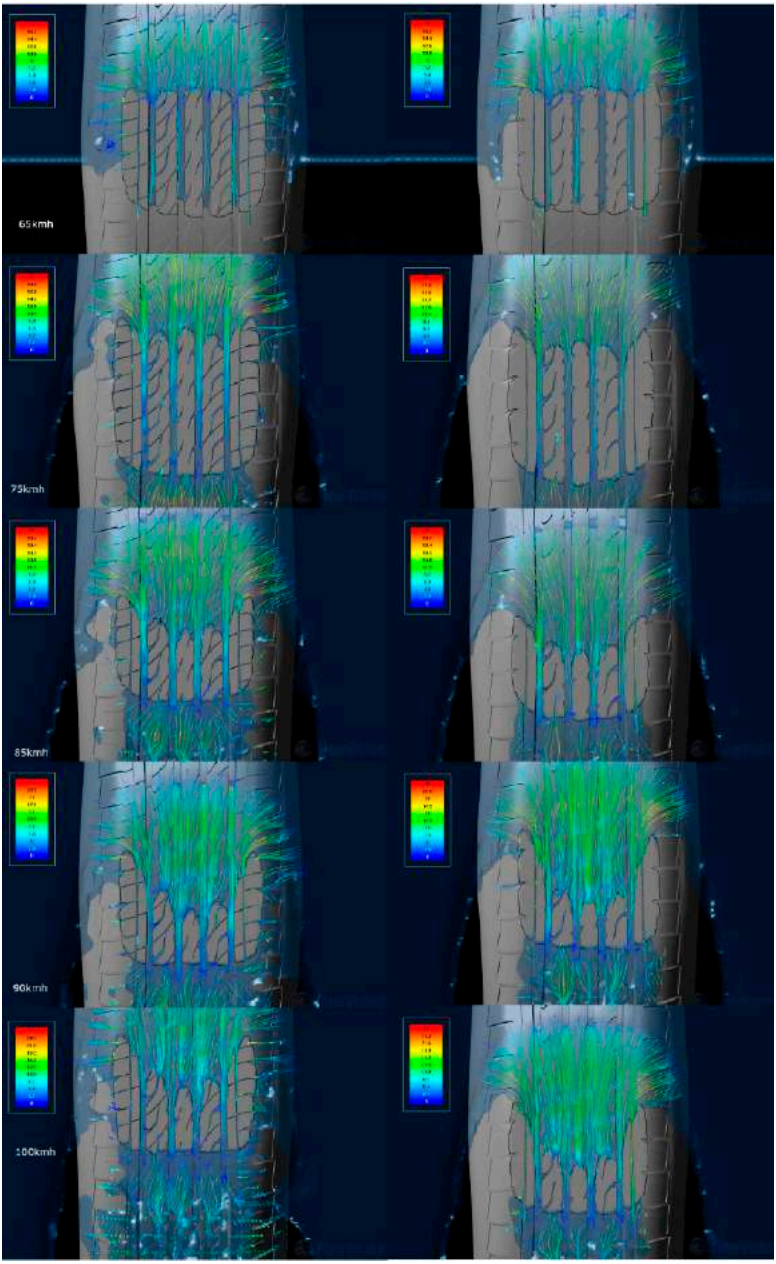


Fig. 17 Streamlinesfor IDT 100 tires with/without lateral grooves at different traveling speeds(coloredby velocity,m/s)

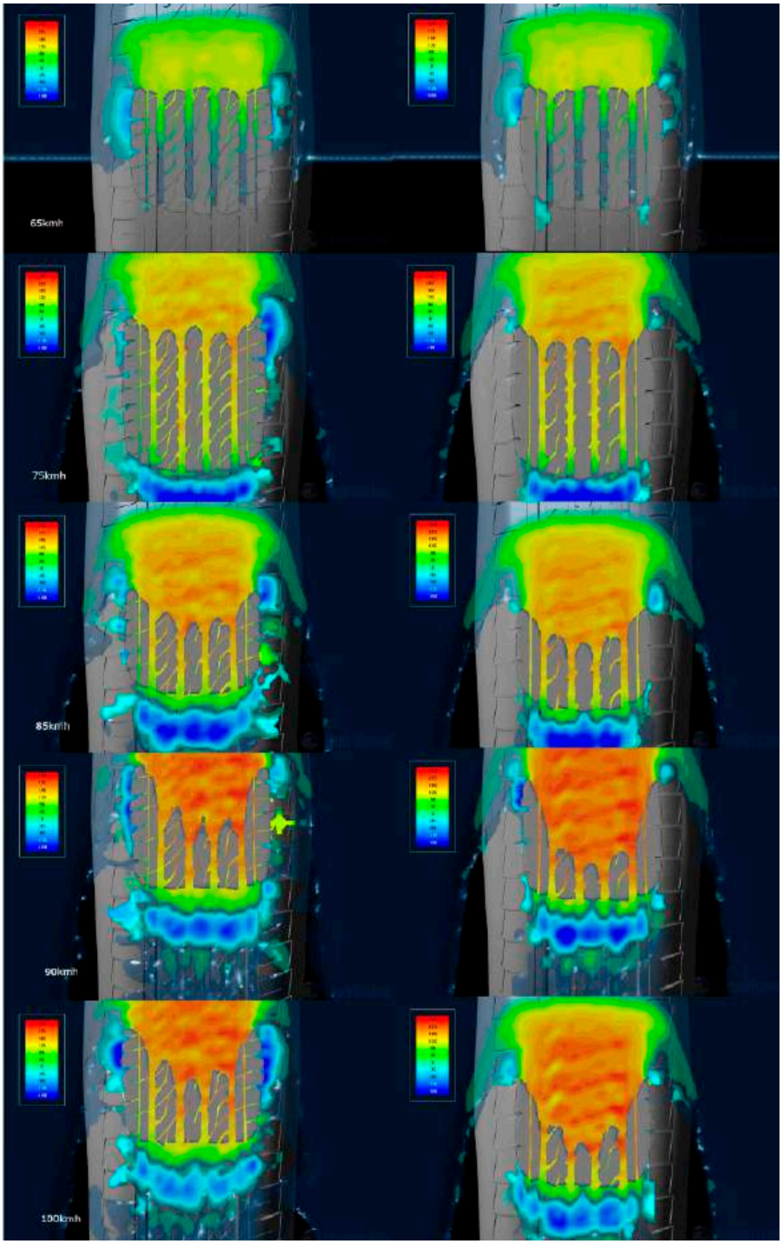


Fig. 18 Hydrodynamic pressurefor IDT 100 tires with/without lateral grooves at different traveling speed (colored byKPa)

4 Conclusion

Anew approach in simulating automobile tire aquaplane is developed. The applied methodology involves a two-way coupling between the FEcode Abaqus/Explicit and the CFD finite-volume code FlowVision, based on Direct Coupling Interface between both codes.We show the dependence of slip ratio of tire on different water depths and traveling speeds.The effect of tread pattern on aquaplaning is also simulated;the results show that increased number of lateral grooves is effectively increasing the critical aquaplaning speed of tire.

REFERENCES

[1] A. Lie, et al., “The effectiveness of ESC in reducing real life crashes and injuries,” Traff. Inj. Prev. 7, 2006, pp.38–43.

[2] J. Strandroth, M. Rizzi, M. Olai, A. Lie, C. Tingvall, “The effects of studded tires on fatal crashes with passenger cars and the benefits of electronic stability control (ESC) in Swedish winter driving,”Accident Analysis and Prevention, Vol.45, 2012, pp.50-60.

[3] M. Matilainen, A. Tuononen, “Tyre contact length on dry and wet road surfaces measured by three-axial accelerometer,”Mechanical Systems and Signal Processing, Vol.52-53, 2015, pp.548-558.

[4] Xin-xin Guo, Chi Zhang, Bu-Xin Cui, Di Wang, James Tsai, “Analysis of Impact of Transverse Slope on Hydroplaning Risk Level,” Procedia-Social and Behavioral Sciences, Vol.96, 2013, pp.2310-2319.

[5] Y. Nakajima, E. Seta, T. Kamegawa, H. Ogawa, “Hydroplaning Analysis by FEM and FVM: Effect of TireRolling and Tire Pattern on Hydroplaning,” FISITA World Automotive Congress, 2000, F2000G382.

[6] E. Croner, H. Bézard, C. Sicot, G. Mothay, “Aerodynamic characterization of the wake of an isolated rolling wheel,” *International Journal of Heat and Fluid Flow*,Vol.43, 2013, pp.233-243.

[7] S. Kim, W. Jeong, Y. Park, S. Lee, “Prediction method for tire air-pumping noise using a hybrid technique,” *J. Acoust. Soc. Am*,Vol.119, 2006, pp. 3799-3812.

[8] Aksenov, A.A., Gudzovsky, A.V., “The software FlowVision for study of air flows, heat and mass transfer by numerical methods,”*Third Forum of Association of Engineers for Heating, Ventilation, Air-Conditioning, Heat Supply and Building Thermal Physics*,Vol.31-35,1993, pp.22-25.

[9] Aksenov A.A., Dyadkin A.A., Gudzovsky A.V.,”Numerical Simulation of Car Tire Aquaplaning,”*Computational Fluid Dynamics*,1996,pp. 815-820.

[10] CAPVIDIA FlowVision HPC version 3.09.05, 2016user manual.

Table 2 Units and numerical settings. [10]

Notation	Physical quantity	Dimension	Notation	Physical quantity	Dimension
C_p	specific heat=1008.69	$m^2s^{-2}K^{-1}$	Q	sum of the energy of different nature	m^2s^{-2}
F	acceleration of external volume face	ms^{-2}	R_A	universal gas constant= 8.31441	$Jmole^{-1}K^{-1}$
G	gravity acceleration	ms^{-2}	T_{tot}	total temperature	°K
H	total enthalpy	m^2s^{-2}	T_{ref}	reference temperature = 273	°K
K	turbulent energy	m^2s^{-2}	T_{abs}	absolute temperature	°K
L	characteristic length	m	T^n	temperature value at time layer n	°K
M	molar mass=0.02884	$kgmile^{-1}$	μ	molecular dynamic viscosity	$kgm^{-1}s^{-1}$
P	relative pressure	Pa	μ_t	turbulent dynamic viscosity	$kgm^{-1}s^{-1}$
P_{ref}	reference pressure= 101000	Pa	V	relative velocity	m/s
P_{hst}	hydrostatic pressure	Pa	V^n	velocity value at time layer n	m/s
P_{abs}	absolute pressure	Pa	ρ^n	density value at time layer n	kg m^{-3}
P_{tot}	total pressure	Pa	ρ^{n+1}	density value at time layer n+1	kg m^{-3}
Pr_t	turbulent prandtl number	----	ε	dissipation rate of turbulent energy	m^2s^{-2}
P^n	pressure value at time layer n	Pa	β	coefficient of thermal expansion	$°K^{-1}$
P^{n+1}	pressure value at time layer n+1	Pa	∂T	relative local specific	Kgs^{-2}
F	Variable of VOF				