Soutenance d'une thèse de doctorat
De l'Université de Lyon
Opérée au sein de l'INSA Lyon

La soutenance a lieu publiquement

Candidat	M. CHEN Bo
Fonction	Doctorant
Laboratoire INSA	LaMCoS
Ecole Doctorale	ED 162 : MÉCANIQUE, ÉNERGÉTIQUE, GÉNIE CIVIL, ACOUSTIQUE
Titre de la thèse	« Approches de coques en efforts résultants pour la simulation de la mise en forme des renforts textiles de composites »
Date et heure de soutenance	21/04/2022 à 13h30
Lieu de soutenance	Amphithéâtre Emilie du Châtelet (Bibliothèque Marie Curie) (Villeurbanne)

Composition du Jury

Civilité	Nom	Prénom	Grade / Qualité	Rôle
M.	BINETRUY	Christophe	Professeur	Examinateur
MME	KÄRGER	Luise	Associate Fellow	Rapporteur
M.	WANG	Peng	Professeur	Rapporteur
M.	NAOUAR	Naim	Chargé de Recherche	Examinateur
M.	BOISSE	Philippe	Professeur	Directeur
M.	COLMARS	Julien	Maître de Conférences	Examinateur

Résumé

Numerical simulation of the textile composite reinforcement plays a critical role in improving the manufacturing quality, reducing the manufacturing cycle, and manufacturing cost. The thickness of the textile composite reinforcement is significate smaller than the other two dimensions, which make it possible to make the simulation using shell element. However, the possible slippage between the fibers gives the material a very specific behavior: the bending behavior is independent of the membrane behavior. The classical shell theory cannot be directly adopted. A stress resultant shell approach is proposed for the simulation of the textile composite reinforcement. This approach introduces an independent bending stiffness to decouple bending behavior with membrane behavior. The stress resultants and stress moments are related to membrane strains and curvatures by rate constitutive equations (hypoelastic laws). Different types of forming experiments under different boundary conditions are conducted to make the comparison between simulation and experiment. The comparison shows the effectiveness and correctness of the proposed approach in predicting the textile composite forming. This approach is implemented in the commercial software ABAQUS and can be used by all users of this software.