Toward the prediction of turbulent boundary layers using a coupled RANS-LES method

By J. Schlüter†

The prediction of turbulent boundary layers using time-resolved methods such as Large-Eddy Simulations (LES) is associated with high computational costs. One possible way to alleviate the stringent restrictions on these methods is to introduce hybrid methods using LES away from the wall, and the Reynolds-Averaged Navier-Stokes (RANS) approach in the near-wall region. A transition must take place between both domains and usually, the challenge is to convert the modeled turbulence in the near-wall region to time-resolved turbulence in LES. An approach is presented below that allows the issue of conversion be circumvented. The LES domain reaches down to the wall and creates an overlap region with the RANS domain. In the overlap region, the LES mean flow solution is driven to the RANS solution using virtual body forces. This allows that the LES domain develops naturally resolved turbulence, while the highly resolved RANS mesh provides a more accurate prediction of the near-wall region without restricting the global time step of the LES solver. The following study presents the motivation, approach and some preliminary results using this approach.

1. Motivation

The motivation of this project stems from the nescessity to investigate turbulent boundary layers for the potential to reduce surface drag. Interest is growing in the study of surface modifications for that effect. Experimental findings suggest that dimples and chevrons have the potential to alter the turbulent structure of the boundary layer in such a way that surface drag is reduced (Carpenter, 1997; Sirovich & Karlsson). These findings were presented so far phenomenologically, without a systematic approach to explain and understand this effect.

Numerical methods are advantageous in assessing in detail the production and destruction of vortical structures by the surface modifications within the boundary layer. Here, the investigation of this phenomenon will rely on Large-Eddy Simulations (LES) and the traditional Reynolds-Averaged Navier-Stokes (RANS) approach. RANS simulations are the most commonly used in engineering CFD. However, since most RANS turbulence models have difficulties in adapting to complex flow phenomena, these are constantly modified to adjust to changing needs. On the other hand, LES resolves the large-scale structures in time and space and is much more versatile in computing a wide variety of turbulence phenomena without an *a priori* knowledge of the flow.

To demonstrate the ability of LES methods in regard to this issue, a series of computations has been performed examine at a channel flow with and without a dimple. LES will help identify large-scale structures, such as bursts, and will help assess the influence of

† Nanyang Technological University, School of Mechanical and Aerospace Engineering, Aerospace Division

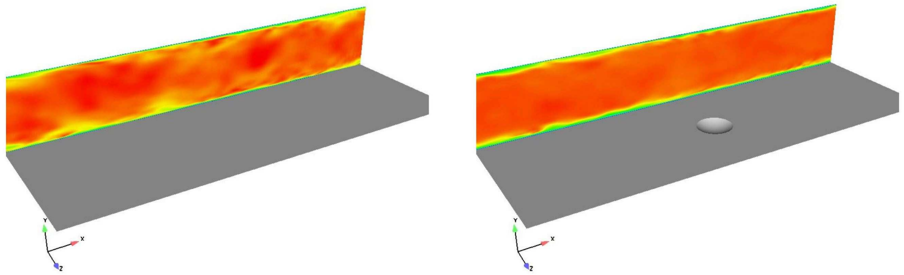

FIGURE 1. Geometry of the channel flow without (left-hand side) and with a dimple (right-hand side). The dimple has a diameter of half of the channel hight and an indentation of 15%.

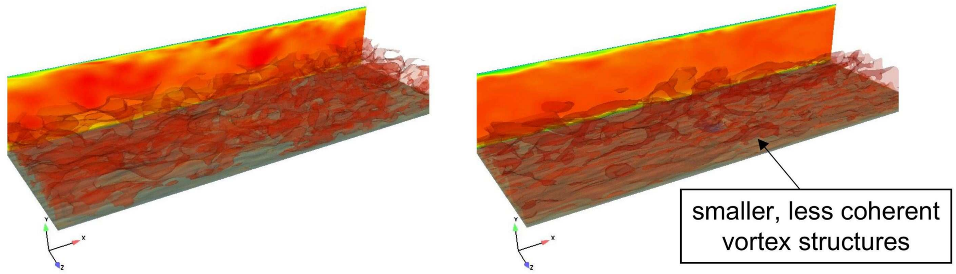

FIGURE 2. Flow visualization of the channel flow without (left-hand side) and with a dimple (right-hand side). Isosurfaces of an instantaneous snapshot of the streamwise velocity component is shown. The back plane shows the axial velocity component as a contour plot. Both cases use identical isosurface and color scale. The boundary layer at the dimpled wall creates smaller, less coherent vortices.

large-scale structures, such as those emanating from the dimples, on the characteristics of the boundary layer. LES has proven successful in the discovery of flow structures in boundary layers (Schlüter *et al.*, 2005a; Wu *et al.*, 2006) and has proven highly accurate in predicting detached flows. Flow visualizations using LES solution sets are powerful tools to identify flow features. Other than water tunnel experiments, LES flow visualizations do not rely on the presence of dye in the flow structure under investigation. Furthermore, LES is able to visualize the flow even at high Reynolds-numbers.

As a test case, a shallow dimple with 15% indentation was selected. Preliminary experimental results suggest that shallow dimples would be able to achieve the desired drag reduction result. Figure 1 shows the geometry of the two test cases: the reference test case of the channel with a smooth wall and the case with a dimple. The Reynolds-number of the channel flow is Re_τ=1000. The diameter of the dimple is half of the channel height H.

The mesh is resolved with about 2 million mesh points and refined around the dimple. The mesh resolution in both the dimpled channel wall, as well as the smooth channel wall is identical, which means that even the smooth wall channel has a mesh refinement at the location where the dimple would be. This refinement was created because in a flow visualization regions of mesh refinement may suggest a smaller vortex structure in this location. The channel domain is eight channel heights H long and four H wide. The upstream and downstream, as well as the lateral boundaries are periodic. A constant body force drives the flow.

The LES flow solver used for this study is the CDP code developed at the Center for Turbulence Research (CTR) at Stanford (Ham *et al.* 2003; Moin & Apte). The filtered momentum equations are solved on a cell-centered unstructured mesh and are second-order accurate. An implicit time-advancement is applied. The subgrid stresses are modeled with a dynamic procedure (Germano *et al.*, 1991).

Figure 2 presents a visualization of the two flows. The isosurfaces of the axial velocity component are shown. The levels of isosurfaces, as well as the color scale in the back plane of the channel are identical for the reference case and the dimple case. Note that the structures in the case with the dimple are smaller and less coherent. Also note at the back plane of the channel the color scale for the reference case shows much more variation.

The assumption from this flow visualization is that the dimple creates small-scale turbulence that breaks up larger structures. As such, it would act in a similar fashion as a Large-Eddy Break-Up (LEBU) device. The challenge is finding an adequate dimple geometry that does not add more drag by creating additional turbulence than it reduces by destroying large-scale turbulence.

Before a comprehensive understanding can be achieved, this topic needs further research.

However, in order to assess the effect of surface modifications on drag reduction, a number of parameter studies have to be performed. Since airplanes are the ultimate recipients of this research, simulations must be done at high Reynolds numbers. Hence, it is useful to investigate methods that reduce computational costs but still provide accurate prediction of the structure of the turbulent boundary layer.

2. Hybrid RANS-LES approach

Time-resolved flow simulations, such as LES and Direct Numerical Simulations (DNS), of turbulent boundary layers usually involve high computational costs because the turbulent length and time scales are very small in the near-wall region. The necessity of LES and DNS to resolve these scales in time and space results in very fine meshes in the near-wall region, and accordingly small time steps. For high Reynolds-number flows, this restriction quickly becomes prohibitive.

One possible way to circumvent this problem is the use of a hybrid method combining the traditional RANS approach with LES. The RANS method is used in the near-wall region, while LES is used in the outer regions of the boundary layer, where the structures are larger.

This approach has been mainly used in the framework of Detached Eddy Simulations (DES) (Spalart, 2000). Here, within one solver, a model is created that slowly converts the subgrid model from a RANS model to an LES model. Multiple criterions have been proposed regarding when to switch from RANS to LES, but no universal criteria has emerged yet.

DES methods also encounter the problem of transitioning from the RANS region (where most of the turbulence is modeled) to the LES region (where the turbulence is resolved in time and space). This is usually done by choosing a small RANS time step equal to that of the LES domain. As a result, some turbulent structures develop in the RANS domain, however these structures do not necessarily need to correspond well to the ones observed in LES or DNS, due to the different modeling.

We propose here to circumvent the transitioning between LES and RANS by using a

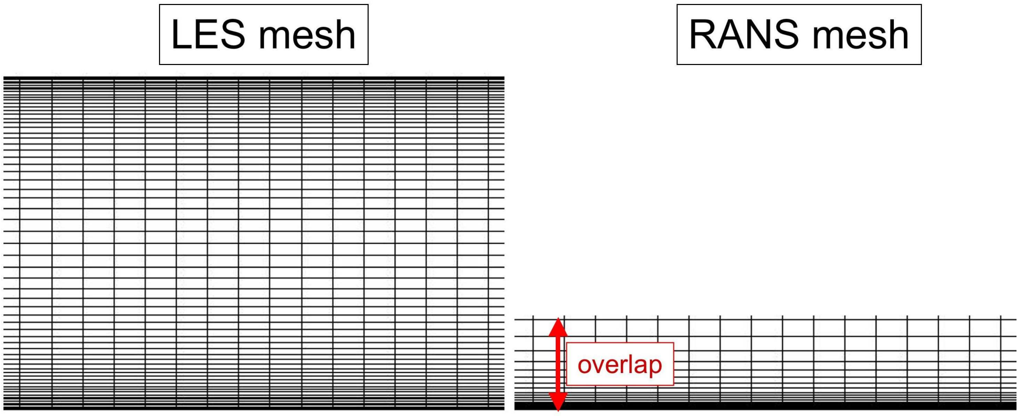

Figure 3. Computational domains of LES and RANS.

zonal approach. In this approach, we use two separate domains, one for RANS and one for LES. The RANS domain covers the volume close to the wall. The LES domain covers the entire domain, including the RANS domain, creating a region of overlap (Figure 3). The advantage here is that the LES domain does not necessarily need to resolve all turbulent structures close to the wall. The RANS domain needs to be finer than that of the LES resolving the laminar sublayer. The advantage is that the LES solution in the overlap region will naturally develop turbulence. The poor resolution of the LES domain will result in an error in the prediction of the mean flow field in the LES domain, which can then be compensated for by the RANS solution. The LES mean flow field will be corrected by a virtual body force that drives the LES mean flow solution to the RANS prediction. This virtual body force technique has been used previously with success.

In the final application, we will be using two separate solvers, one for the LES and one for the RANS. This has a number of advantages, one of which is that the time step of both solvers not need be identical. The time step of the RANS flow solver can be much larger than that of the LES solver, reducing the actual computational costs.

The RANS and the LES solvers communicate flow solutions over the coupling software module CHIMPS.

3. Coupling approach

While previous LES-RANS hybrid approaches, such as Detached-Eddy Simulations (DES) (Spalart, 2000) and Limited-Numerical Scales (LNS) (Batten *et al.*, 2002), combine LES and RANS in a single flow solver, the approach of coupling two existing flow solvers has significant advantage: it builds upon the experience and validation applied into the individual codes during their development. Also, simulations can be run in different domains at different time steps. Both LES and RANS require a distinct set of numerical algorithms and models to work efficiently and accurately. The integration of both approaches into a single solver is often tiresome.

Presented here is a different approach to address this problem: We keep the solvers separate. They run simultaneously and exchange only the necessary information at the interfaces. This allows each solver to use the best and most accurate methods for the solution of its problem, while the interaction with other solvers allows for an approach to very complex engineering applications.

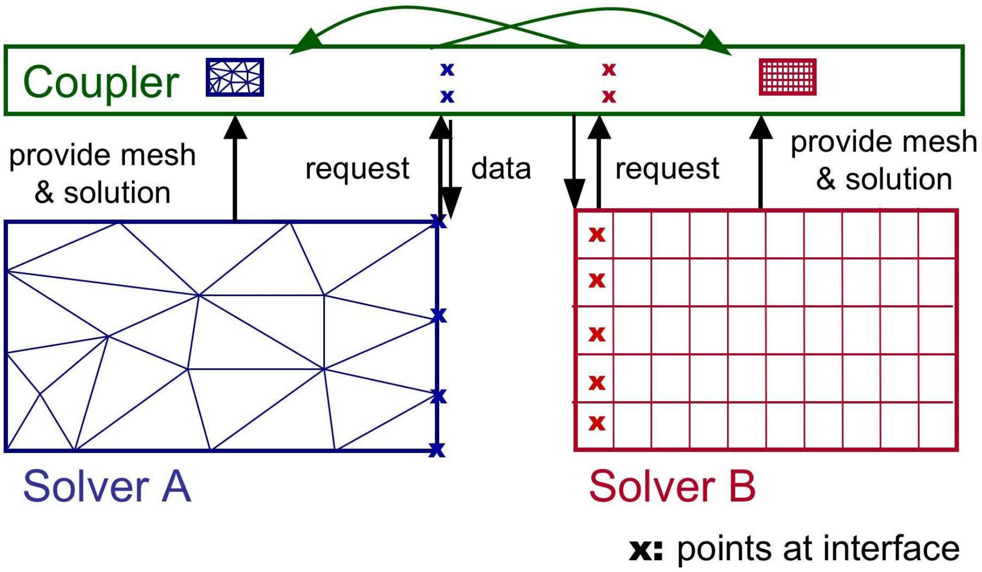

FIGURE 4. CHIMPS approach: solvers communicate location of their interface points and their mesh and solution to the coupler. The coupler determines how to provide information to the solver at the interface nodes.

For the instance of LES-RANS hybrids, the coupling approach allows for the use of the most appropriate approach for each zone in a domain. For example, one part can be computed using a compressible structured multi-block RANS solver and the other by a low Mach-number unstructured LES solver. This approach has been successfully applied to a variety of flow phenomena (Schlüter *et al.*, 2005b), including complex real-world engineering applications (Schlüter *et al.*, 2005c).

Previous approaches to couple solvers were based on a pure MPI approach (Shankaran *et al.*, 2001, Schlüter *et al.*, 2005b). In this approach, MPI is used to establish a direct communication between the solvers. This requires that in each of the solvers, algorithms be implemented that perform the tasks associated with the coupling.

During the actual computation, the solvers exchange interface data directly. After each time step, the solvers interpolate their own solution on the interface nodes of the other solver and send the information to their peers.

The disadvantage of this approach is that it takes some effort to implement the communication algorithms into a new solver. Since each MPI command in one solver requires a corresponding MPI command in the other solver, the implementation may be tedious and prone to errors. Furthermore, the search and interpolation routines must be implemented in each solver separately.

In order to improve the interoperability of the solvers, we decided to approach the coupling in a different manner. Instead of implementing all coupling routines (communication, search, and interpolation) in all solvers separately, we have developed a separate software module that performs these tasks and facilitates the coupling process. The goal here is to remove the workload, especially the search, interpolation, and communication routines, from the solvers. The solvers communicate only with the coupler software (Figure 4). The coupler performs all searches and interpolations. In order to perform these tasks, the coupler requires knowledge of the meshes and of the solvers' solutions.

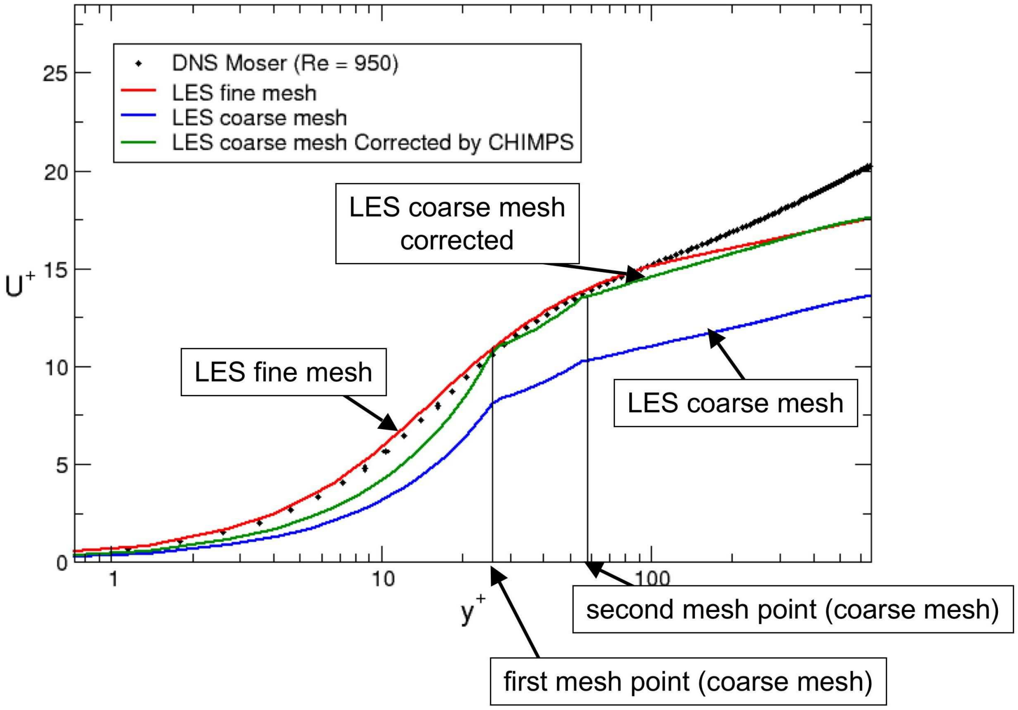

FIGURE 5. Computational domains of LES and RANS.

The coupling software module has been developed is called **C**oupler for **H**igh-Performance **I**ntegrated **M**ulti-**P**hysics **S**imulations (CHIMPS) (Schlüter *et al.*, 2005d; Alonso *et al.*, 2006), and is based on the script language python. This script language, together with its parallel version pyMPI, allows for the simplification of the communication between the solvers and CHIMPS. Instead of defining MPI calls, python functions are defined allowing for more freedom in the implementation. The communication is then handled like a function call with arguments, with the data being passed in the argument list.

An extension of CHIMPS is a Fortran-only version. This version does not require python and relies on the solvers and the coupling software to be compiled as libraries linked within one master routine.

4. Coupled RANS-LES for turbulent boundary layers

In order to test this approach for turbulent boundary layers, we have performed a series of computations on a turbulent channel flow to compare the proposed hybrid RANS-LES with the results of a fully resolved LES computation.

The mesh size for the fully resolved LES computation is 64 points in crosswise direction, 64 points in spanwise direction and 128 points in streamwise direction. The wall distance of the first point is $y^+ = 1$. The Reynolds-number is equal to $Re_\tau = 1000$. This resolution provides a nearly fully resolved DNS.

The results of this reference test case are shown in Figure 5. The results compare well, especially in the near-wall region with the DNS data from Moser *et al.*.

In a second simulation, we consider a poorly resolved LES computation. In this case the mesh size is 32 points in crosswise direction, 64 points in spanwise direction and 128 points in streamwise direction. The wall distance of the first point is $y^+ = 100$, and is hence underresolved. The driving force term that drives the flow through the periodic channel remains the same.

As is shown in Figure 5 the results in the near-wall region are poor. As a consequence of the inaccurate prediction of the shear stresses near the wall, the main flow field and the bulk velocities adjust to an inaccurate prediction.

In order to demonstrate the advantage of the coupled RANS-LES approach, we will now use the time-averaged results from the fully resolved computation as the "exact" RANS solution. We chose to use this data instead of an actual RANS simulation in order to deliberately exclude inaccuracies that may arise from a RANS solver. We use the coupling software CHIMPS to provide the mean flow field solution as a RANS solution to the underresolved LES computation. The LES solver then uses virtual body forces to drive the LES mean flow solution toward the provided RANS solution in a region $0.2H$ close to the wall.

As a result (see Figure 5), the mean flow field in the boundary layer adjusts to the desired values. Note that not only the flow velocity in the wall-near region matches that of the fully resolved LES, but also toward the channel center. Here, the accurate prediction of the shear stress at the wall results in a more accurat.e prediction outside the near-wall region.

No results concerning the computational costs have yet been derived from the computations. However, looking at the coarser LES mesh it can be clearly seen that the computational costs are much lower than that of the resolved LES. Since the mesh has been coarsened by a factor of 100, in a compressible flow solver this would mean that the time step that is dependent on the speed of sound and the smallest cell in the domain can be chosen 100 times larger compared to the resolved mesh.

5. Conclusions and future work

We have shown, in this study how surface modifications can alter the vortex structure of a turbulent boundary layer. Since these surface modifications are to be used for the manipulation of drag that are in the order of magnitude of a few percent only, very accurate simulation tools are required to predict and examine this phenomenon. Yet, efficient methods are required due to the area of application at high Reynolds-numbers.

We have proposed a method to use a hybrid RANS-LES approach, where the RANS domain predicts the flow in the near-wall region, and the LES domain extends through the entire domain, including the near-wall region, albeit with an underresolved mesh. The advantage of this approach is that the RANS solution is able to predict the near-wall region efficiently, yet the turbulence in the LES domain is generated naturally with its own domain. Vortex structures generated at the described wall roughness modifications are especially better captured. The LES mean flow field is corrected toward the RANS solution using a body force technique.

Preliminary results of this method on a turbulent channel flow show the potential to predict turbulent boundary layers.

Future research will intensify the work in this area using two solvers, one for the RANS and one for the LES domain. Furthermore, the LES and the hybrid RANS-LES approach need to be validated with experimental data on wall surface modifications. These experiments are currently in the planning stages at other institutions and will provide further information on the effect of dimples on boundary layers.

6. Acknowledgements

We wish to thank Dr. Frank Ham for the support with the flow solver CDP and Dr. Wu and Dr. You for discussions on the turbulent channel flow simulations.

We thank the Center for Turbulence Research for its support during the CTR Summer Program.

REFERENCES

Alonso, J. J. , Hahn, S. , Ham, F. , Herrmann, M. , Iaccarino, G. , Kalitzin, G. , LeGresley, P. , Mattsson, K. , Medic, G. , Moin, P. & Pitsch, H. 2006 CHIMPS: A high-performance scalable module for multi-physics simulations. *AIAA* 2006-5274, January 2006.

P. Batten, U. Goldberg & S. Chakravarthy, 2002 LNS-An approach towards embedded LES. *AIAA* 2002-0427, January 2002.

Carpenter, P. 1997 The right sort of roughness. *Nature*, 388, 713–714.

Germano, M., Piomelli, U., Moin, P. & Cabot, W., 1991 A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids* A (3), 1760–1765.

Ham, F., Apte, S., Iaccarino, G., Wu, X., Herrmann, M., Constantinescu, G., Mahesh, K. & Moin, P. 2003 Unstructured LES of reacting multiphase flows in realistic gas turbine combustors. *CTR Annual Research Briefs*, pages 139–160, 2004. Center for Turbulence Research, Stanford.

Moin, P. & Apte, S. 2004 Large-eddy simulation of realistic gas turbine combustors. *AIAA* 2004-0330, January 2004.

Schlüter, J. U. , Pitsch, H. & Moin, P. 2005 Outflow conditions for integrated large-eddy simulation/Reynolds-averaged Navier-Stokes simulations. *AIAA Journal*, 43(1) 156–164.

Schlüter, J. U. , Wu, X. & Pitsch, H. , 2005 LES of a separated plane diffuser. *AIAA* 2005-0672, January 2005.

Schlüter, J. U. , Wu, X. , Kim, S. , Alonso, J. J. & Pitsch, H. , 2005 A framework for coupling Reynolds-averaged with large eddy simulations for gas turbine applications. *Journal of Fluids Engineering*, 127, 4, 8006-815.

Schlüter, J. U. , Wu, X. , Kim, S. , Alonso, J. J. & Pitsch, H. , 2005 Integrated Simulations of a compressor/combustor assembly of a gas turbine engine. *ASME paper*, GT2005-68204, ASME Turbo Expo 2005.

Schlüter, J. U. , Wu, X. , v. d. Weide, E. , Hahn, S. , Alonso, J. J. & Pitsch, H. , 2005 Multi-code simulations: A generalized coupling approach *AIAA* 2005-4997, June 2005.

Shankaran, S. , Liou, M.-F. , Liu, N.-S. , Davis, R. & Alonso, J. J. , 2001 A multi-code-coupling interface for combustor/turbomachinery simulations. *AIAA* 2001-0974, January 2001.

Sirovich, L. & Karlson, S. 1997 Turbulent drag reduction by passive mechanisms. *Nature*, 388, 753-755.

Spalart, P. R. , 2000 Trends in turbulence treatments. *AIAA* 2000-2306, June 2000.

Wu, X. , Schlüter, J. U. , Moin, P. , Pitsch, H. , Iaccarino, G. & Ham, F. 2006 Computational study on the internal layer in a diffuser. *Journal of Fluid Mechanics*, 550, 391-412.