



Centrum voor Wiskunde en Informatica

REPORT*RAPPORT*

MAS

Modelling, Analysis and Simulation



Modelling, Analysis and Simulation

FLECS, a flexible coupling shell, application to
fluid-structure interaction

M. Nool, E.J. Lingen, A. de Boer, H. Bijl

REPORT MAS-E0624 DECEMBER 2006

Centrum voor Wiskunde en Informatica (CWI) is the national research institute for Mathematics and Computer Science. It is sponsored by the Netherlands Organisation for Scientific Research (NWO). CWI is a founding member of ERCIM, the European Research Consortium for Informatics and Mathematics.

CWI's research has a theme-oriented structure and is grouped into four clusters. Listed below are the names of the clusters and in parentheses their acronyms.

Probability, Networks and Algorithms (PNA)

Software Engineering (SEN)

Modelling, Analysis and Simulation (MAS)

Information Systems (INS)

Copyright © 2006, Stichting Centrum voor Wiskunde en Informatica
P.O. Box 94079, 1090 GB Amsterdam (NL)
Kruislaan 413, 1098 SJ Amsterdam (NL)
Telephone +31 20 592 9333
Telefax +31 20 592 4199

ISSN 1386-3703

FLECS, a flexible coupling shell, application to fluid-structure interaction

ABSTRACT

Numerical simulations involving multiple, physically different domains can be solved effectively by coupling simulation programs, or solvers. The coordination of the different solvers is commonly handled by a coupling shell. A coupling shell synchronizes the execution of the solvers and handles the transfer of data from one physical domain to another. In this paper, we introduce Flecs, a flexible coupling shell, designed for implementing and applying an interface for multidisciplinary simulations with superior accuracy. The aim is not to achieve the best possible efficiency or to support a large feature set, but to provide a flexible platform for developing new data transfer algorithms and coupling schemes.

2000 Mathematics Subject Classification: 68N99; 74F10.

1998 ACM Computing Classification System: G4

Keywords and Phrases: Coupling Shell; Fluid-Structure Interaction; parallelism bu MPI-2

Note: Funding for this work was provided by the National Facilities Foundation (NCF), under project number NRG-2005.03

FLECS, a Flexible Coupling Shell

Application to Fluid-Structure Interaction *

Margreet Nool¹, Erik Jan Lingen², Aukje de Boer³, and Hester Bijl³

¹ CWI, Department of Computing and Control, Amsterdam, The Netherlands

² Habanera Software company, Delft, The Netherlands

³ TU Delft, Faculty of Aerospace Engineering, Delft, The Netherlands

Abstract. Numerical simulations involving multiple, physically different domains can be solved effectively by coupling simulation programs, or *solvers*. The coordination of the different solvers is commonly handled by a *coupling shell*. A coupling shell synchronizes the execution of the solvers and handles the transfer of data from one physical domain to another. In this paper, we introduce FLECS, a flexible coupling shell, designed for implementing and applying an interface for multidisciplinary simulations with superior accuracy. The aim is not to achieve the best possible efficiency or to support a large feature set, but to provide a flexible platform for developing new data transfer algorithms and coupling schemes.

1 Introduction

Fluid-Structure Interaction (FSI) considers coupled fluid-solid problems, characterized by the interaction of fluid forces and structural deformations, which occur in many applications in industry and science. Nowadays, the simulation of FSI becomes more and more important, since future structures become lighter and more flexible and can be applied, e.g., to reduce the load on turbine blades, or, to reduce the noise on cars. Such applications require a real interdisciplinary approach, that can deal with complex physical models and very different scales.

The Faculty of Aerospace Engineering of the Delft University of Technology has started a project to develop a generic, open-source coupling shell, named FLECS [3], that can be used to join two or more arbitrary solvers. FLECS should provide an innovative combination of high order coupling in space and time. Moreover, to improve the accuracy and the efficiency of the computation, multilevel acceleration techniques for the coupling process [2], and fast prototyping and parallelization techniques will be supported.

The majority of coupling shells are embedded subprograms that have been developed for coupling two specific solvers. One exception is the coupling library MPCCI (Mesh based Parallel Code Coupling) [5], which can be used as a separate program. Although MPCCI is relatively easy to use and provides

* Funding for this work was provided by the National Computing Facilities Foundation (NCF), under project numbers NRG-2005.03.

many advanced features, it is less suitable for a scientific research community that is aimed at developing new data transfer algorithms. Numerical acceleration algorithms, like Krylov and multilevel methods - urgently required for efficiency - are not incorporated. Moreover, since MPCCI only provides the binary code, the user can not modify the implementation schedule of MPCCI.

In addition to accurate coupling in space, we want to reduce the partitioning errors in time by using specially designed high-order time integration methods [6]. It is our goal for FLECS to support solvers that run on parallel computers, in order to make FLECS suitable for large applications. In particular, FLECS must be able to deal with data sets that have been distributed over multiple parallel processes. In addition, FLECS should support the implementation of parallel data transfer algorithms. At the moment, experiments are limited to sequential solvers; each solver, running on its own processing unit, deals with one particular physical domain. The coupling server, running on a third processing unit, takes care of the exchange of data between the solvers. Since FLECS has to deal with separate solver processes, that have been started independently, it is not possible to use `MPI-1` for exchanging data between those processes. For that reason, FLECS is based on `MPI-2` [4].

The remainder of the paper is structured as follows. In Sect. 2, we describe the importance of coupling two solvers properly to solve interdisciplinary problems in an efficient and accurate way. Sect. 3 gives an overview of the design of FLECS. Through the help of a test problem, Sect. 4 illustrates how FLECS can be used. And finally, Sect. 5 contains some conclusions and future plans.

2 Coupling methods

Interdisciplinary problems can be solved in two ways. In the first way, the so-called *monolithic* approach, a new dedicated solver is developed that solves the whole system at once. Major advantage is that the solver can be optimized for the specific problem. Development of such a complex, entirely new solver, will take an enormous effort, while there already exists many highly efficient and accurate solvers for the separate domains. The other way, called the *partitioned* approach, is to reuse monodisciplinary solvers, that have been developed and tuned for tens of years. In that case, each physical system is solved individually and interaction effects are treated as external conditions. A disadvantage of this approach is that the coupling algorithm is not as straightforward as it looks. Without much care the accuracy of the coupled problem easily reduces to first-order in time, irrespective of the order of the separate solvers.

FLECS provides an efficient coupling interface for partitioned computation of multidisciplinary problems. The design of FLECS allows all kinds of data transfer algorithms to couple different domains in space and time. Numerical acceleration techniques, like multigrid, can be incorporated, too.

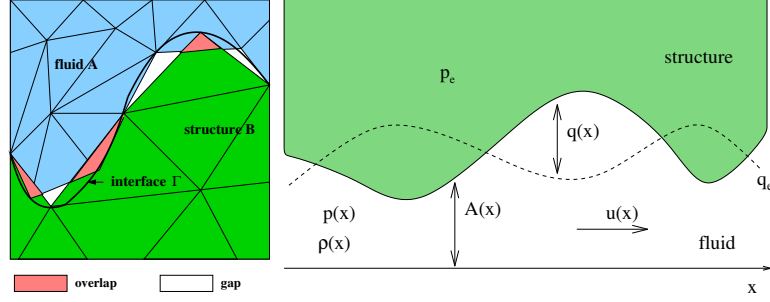


Fig. 1. (a) Non-matching grids in 2D and (b) Configuration of the quasi-1D test problem

2.1 Non-matching grids

If different grid generators are used for both domains, the mesh interface may not only be non-confirming (nodes at the interfaces match, same discretization and same interpolation order on both discrete interfaces), but also non-matching (cf. Fig. 1(a)) in the way that there are gaps and/or overlaps between the meshes. We remark, that generating matching grids is most of the time not desirable, because, in general, the simulation on one physical domain requires a much finer grid than the simulation on the other one. In the remainder of this paper, we consider FSI rather than some arbitrary physical domain interaction.

De Boer et al. [1] gives a detailed study of data transfer methods, and FSI simulations are performed on non-matching meshes. As coupling method in this paper, we take the radial basis function method (RBF) with large compact support. This method, favored by [1], because of its accuracy and efficiency, does not need orthogonal projection or search algorithms. The coupling between fluid and structure equations arises from the dynamic and kinematic boundary conditions (BC) at the fluid-structure interface. The BC for the displacement on the continuous fluid-structure interface Γ , given by $dx_A = dx_B$, where dx denotes the displacement on either the flow or structure side of the interface. The displacements of the flow points have to be predicted once the displacements of the structure points have been determined. The discrete version of the BC can be formulated as

$$d\mathbf{x}_A = \mathbf{H}_{AB} d\mathbf{x}_B,$$

where $\mathbf{H}_{AB} \in \mathbb{R}^{N_A \times N_B}$ is the transformation matrix prescribed by the RBF method. The numbers N_A , the number of flow, and N_B , the number of structure points on the fluid-structure interface, are usually very small compared to the total number of structure and flow points. Analogously, the discrete version of the BC for the pressure forces leads to

$$\mathbf{p}_B = \mathbf{H}_{BA} \mathbf{p}_A,$$

where the transformation matrix \mathbf{H}_{BA} is of size $N_B \times N_A$. The computation of \mathbf{H}_{BA} involves the inversion of small matrix of size $N_A \times N_A$.

2.2 Coupling algorithm

We consider, as an example of use, a quasi 1-D channel with a flexible curved wall as shown in Fig. 1(b). The main velocity of the compressible flow is in the x -direction and the structure is modeled as a membrane. The diameter of the tube may vary due to a pressure difference between the pressure in the flow and in the wall. For more details on this test problem, we refer to [1].

The simulation of the compressible flow and the membrane is solved effectively by coupling two solvers. The solvers exchange data to take into account the effects on the other domain. Starting at time t^n , each solver computes the solution at time $t^n + \Delta t^n$ on its own particular domain. In general, Δt^n will be determined by the flow solver. The following steps are carried out to obtain the solution at t^{n+1} from the solution at t^n :

- step 1.** compute transformation matrices \mathbf{H}_{AB}^n and \mathbf{H}_{BA}^n
- step 2.** obtain the pressure on the interface points $\mathbf{p}_{B,\gamma}^n = \mathbf{H}_{BA}^n \mathbf{p}_{A,\gamma}^n$
- step 3.** calculate the displacements of the structure \mathbf{dx}_B^{n+1} from the structure equations using the old value of the pressure \mathbf{p}_B^n
- step 4.** use the coupling method to compute the displacements of the tube wall $\mathbf{dx}_{A,\gamma}^{n+1} = \mathbf{H}_{AB}^n \mathbf{dx}_{B,\gamma}^{n+1}$
- step 5.** calculate the new pressure $\mathbf{p}_{A,\gamma}^{n+1}$ from the fluid equations with the new displacements of the tube \mathbf{dx}_A^{n+1} .

The subscript γ denotes that the operations are only performed on data at the interface points. The steps to gather and scatter the data on the interface points have been omitted. The computation of \mathbf{H}_{AB}^n and \mathbf{H}_{BA}^n requires the coordinates on the same time t^n . In Sect. 4, we will return to this example of use.

3 Design Overview

FLECS is decomposed into a *client library* that is to be called from the solver programs, and a *coupling server*, in short *server*, that coordinates the execution of the solvers, takes care of the coupling of the domains and handles the transfer of data between the solvers. In the simplest case the server comprises a single process (as in Fig. 2) that executes the transfer algorithm sequentially.

To limit the complexity of the server, it can only couple two solvers at a time. However, one can couple a solver to two other solver processes by starting a second server.

3.1 The Client Library and its Usage

The client library provides subroutines for establishing a connection with the server; for describing the geometry of the coupling interface; for describing the data that are to be transferred to and from the server; for sending data to the server; and for receiving data back from the server. If a solver program comprises multiple parallel processes, then each process will be linked to its

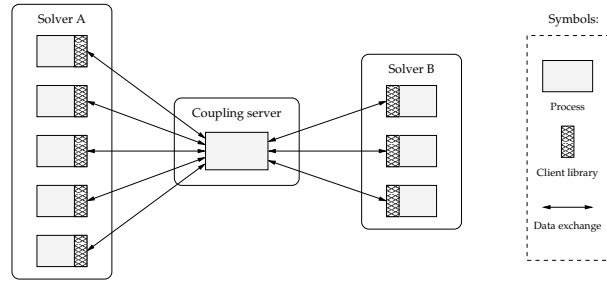


Fig. 2. Schematic representation of FLECS

own copy of the client library, and will establish a separate connection with the server.

Each solver program can be started independently, using its standard start-up procedure; there is no need to change the structure of the solver program. In fact, one only needs to extend the solver program with a small number of subroutine calls to the client library. For an overview of the functions exported by the FLECS client library, see Table 1.

To start a coupled simulation, both solvers set up a connection to the server by calling the client function `FLECS_Connect`. Then the solvers inform the server about the data to be transferred between them. Therefore, one or more *point sets* and *data sets* are created at each side of the interface, and, transferred to the server by calling the functions `FLECS_NewPointSet` and `FLECS_NewDataSet`. The solvers exchange data via the server by calling the functions `FLECS_SendDataSet`, `FLECS_RecvDataSet`, `FLECS_Send` and `FLECS_Recv`. The first pair transfers a data set from one solver to the other, and typically invokes a transformation algorithm on the server. The second pair transfers an arbitrary data array between the solvers, and is particularly used to communicate convergence and time stepping information between both solvers. Both pairs of functions require that a send operation *matches* a corresponding receive operation. The description of a single iteration step of the test problem of Sect. 2.2 can be found in Sect. 4.

3.2 The Coupling Server

The server consists of two parts, as shown in Fig. 3: a *communication and coordination layer*, and a *transfer algorithm*. The communication and coordination layer handles the initialization and finalization of the server; exchanges data between the server and the two solver programs; manages the data structure – including point sets, couplings, and data sets – that have been created by the client library on behalf of the solver programs; and manages the coupling-specific data structures that have been created by a transfer algorithm. Obviously, more than one data set can be associated with a point set. To associate a data set to a particular

Table 1. An overview of the functions exported by the FLECS client library

Initialization functions	
<code>FLECS_Init</code>	Initializes MPI, parses the program names, opens the MPI port, publish the name of the server, save the server address
<code>FLECS_Connect</code>	Connects solver and coupling server
<code>FLECS_NewPointSet</code>	Registers point set on coupling interface with coupling server
<code>FLECS_NewCoupling</code>	Defines coupling between point set on this solver to point set on another solver
<code>FLECS_NewElemSet</code>	Defines element set on coupling interface with coupling server
<code>FLECS_NewDataSet</code>	Defines data set associated with point set
Finalization functions	
<code>FLECS_Disconnect</code>	Disconnects solver from coupling server
<code>FLECS_Shutdown</code>	Calls <code>MPI_finalize</code> and cleans up the allocated memory
<code>FLECS_DelPointSet</code>	Deletes registered point set
<code>FLECS_DelCoupling</code>	Deletes created coupling
<code>FLECS_DelElemSet</code>	Deletes created element set
<code>FLECS_DelDataSet</code>	Deletes defined data set
Data exchange functions	
<code>FLECS_SendDataSet</code>	Sends data set to other solver via coupling server
<code>FLECS_RecvDataSet</code>	Receives <i>transformed</i> data set from other solver
<code>FLECS_Send</code>	Sends <i>arbitrary</i> data array to other solver
<code>FLECS_Recv</code>	Receives <i>arbitrary</i> data array from other solver
Miscellaneous functions	
<code>FLECS_SetCoords</code>	Updates coordinates of registered point set
<code>FLECS_ErrorString</code>	Converts error code to human-readable error message
<code>FLECS_UseElemSet</code>	Use element set

point set, an integer value `pset`, which uniquely identifies a point set, must be involved in the data set message.

The transfer algorithm handles the conversion of a data set from one point set to another point set. This part of the server is based on a plug-in architecture, that makes it easy to implement new transfer algorithms, see also Sect. 4.1. Although these functions have been written in the programming language C, the transfer algorithm itself can be implemented in any programming language. By splitting the server into two parts, one can experiment with different types of transfer algorithms without having to worry about non-essential details such as communication between the server and the solver programs.

4 Execution of Test Problem using FLECS Routines

Again, we consider the quasi-1D test problem described in Sect. 2.2. For simplicity, we assume that flow solver **A** and structure solver **B** have not been parallelized and that each solver process runs on a single processing unit. The

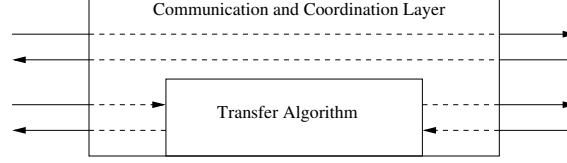


Fig. 3. The coupling server, consisting of a communication and coordination layer and a transfer algorithm. The arrows indicate the flow of data between the coupling server and two solver programs

server Γ becomes a third process which takes care of the communication and the interpolation of the meshes.

In Fig. 4, representing a single iteration, the various steps of the coupling algorithm, as listed in Sect. 2.2, are shown, and, on the solvers **A** and **B** the calls to the client library of FLECS are inserted. We choose, that the flow solver determines the progress of the integration process, i.e., the time step is calculated by the flow solver. In addition, the flow residual R_A^{n+1} controls the complete system. When the solver **A** has computed the next time step Δt^n , its value must be sent via the server Γ to the solver **B** using the pair (FLECS_Send, FLECS_Recv).

Figure 4 illustrates that by breaking the computation of the transformation matrices \mathbf{H}_{AB}^n and \mathbf{H}_{BA}^n (**step 1**) into two separate parts, the server can compute these matrices simultaneously with other operations performed by the solvers. More precisely, no extra wall clock time is needed to compute these matrices. Let us assume that \mathbf{H}_{BA}^n has been calculated in a previous integration step, then the force on the structure can be updated (**step 2**). As a result, we obtain the vector $\mathbf{p}_{B,\gamma}^n$ asked by solver **B**, by means of a call of FLECS_RecvDataSet. Next, solver **B** computes the solution at the structure domain (**step 3**) at time $t^n + \Delta t^n$. We observe that the computation of \mathbf{H}_{AB}^n can be postponed, allowing the solver **B** to start the computation of **step 3** earlier. As a consequence, the computation of \mathbf{H}_{AB}^n can be carried out simultaneously with **step 3**. The vector $\mathbf{dx}_{B,\gamma}^{n+1}$ is needed to carry out **step 4**, and corresponds to calls of FLECS_SendDataSet and FLECS_RecvDataSet on the solvers **B** and **A**, respectively.

Again, by transferring $\mathbf{dx}_{B,\gamma}^{n+1}$ (calling FLECS_SendDataSet) first and then the new point set $\mathbf{x}_{B,\gamma}^{n+1}$ (calling FLECS_SetCoords) more parallelism can be obtained. However, as stated above, the transformation must be applied on updated values of $\mathbf{dx}_{B,\gamma}^{n+1}$. The transformation operation delivers new values $\mathbf{dx}_{A,\gamma}^{n+1}$ to be transferred to solver **A** (calling FLECS_RecvDataSet). Next, the flow solver carries out an integration step from t^n till t^{n+1} (**step 5**). Simultaneously, the matrix \mathbf{H}_{AB}^{n+1} for the flow-structure interface can be calculated using the new point sets $\mathbf{x}_{A,\gamma}^{n+1}$ and $\mathbf{x}_{B,\gamma}^{n+1}$ (**step 1**).

Let ε be some given tolerance, and, let t_e be the end time, then if

$$\text{finish} = R_A^{n+1} < \varepsilon \mid t_n + \Delta t^n \geq t_e$$

is not true, a new iteration step $n+1$ can be started after calculating the new time step Δt^{n+1} . In case of convergence, the process terminates, and solver **A** sends messages to server *I* and solver **B**. This can be carried out by calls to the routines FLECS_Send and FLECS_Recv, followed by calls to FLECS_Disconnect and FLECS_Shutdown to disconnect the connections and to clean up all MPI states.

4.1 The Server Program

The main loop of the server program can look like

```
int main ( int argc, char** argv )
{ flecs_transfer_t transf;
  int result;

  FLECS_InitTransfer ( &transf );
  transf.NewCoupling = NewCoupling;
  transf.DelCoupling = DelCoupling;
  transf.InitPoints = InitPoints;
  transf.SetCoords = SetCoords;
  transf.TransferData = TransferData;

  FLECS_SetTransfer ( &transf );
  FLECS_SetErrorMode ( FLECS_ERRORS_ABORT );
  FLECS_Init ( &argc, &argv );
  FLECS_Connect ( );

  do
  { result = FLECS_MainLoop ( ); }
  while ( ! result );

  FLECS_Shutdown ( ); return 0;
}
```

The do while loop is executed repeatedly until `result` becomes FALSE. The subroutine FLECS_MainLoop *listens* whether there is a message to be received, where after the server copies the data out of the send buffer and will act accordingly. A *message* can be one of the functions listed in Table 1. Assume that the application demands a multigrid approach, and, assume that FLECS_MainLoop receives a message from solver **A** for a new coupling (i.e., solver **A** has called FLECS_NewCoupling). Then the server *I* may expect a similar request by solver **B**, followed by new point sets and associated data sets of both solvers. We remark that as a consequence of this simple approach, illustrated by the above program listing, the FLECS' user does not have to implement the server program, but only adds some simple programs NewCoupling, DelCoupling, InitPoints, SetCoords and TransferData. The latter must include the coupling algorithm. Moreover, the user has to include in his/her solver programs some calls to FLECS routines. Such calls must *correspond*, e.g., FLECS_

Send and FLECS_Recv are appearing in pairs, otherwise an error will be generated.

Obviously, the server can receive messages from both solvers. Beforehand, it is not clear to the server from which solver it receives its next message. Therefore, the function MPI_IPROBE is called to find out whether a queue contains a message or not. In case both queues are empty, the server becomes idle, and, MPI_IPROBE is called again until at least one of the queues is non empty. In case only one queue contains a message, the server accepts that message and acts accordingly. The third possibility corresponds to messages in both queues. In that case the server has to choose between one of the messages. Several strategies are possible to handle this case:

1. alternate the actions received by the solvers **A** and **B** as much as possible
2. consider which message is most 'welcome'. If, e.g., the server has received from solver **B** a new point set, but not of solver **A**, while a request of solver **B** has been arrived for a data set based on a new transformation matrix, than the new point set of **A** will be most welcome.

5 Conclusions and future plans

In this paper, we have introduced FLECS, a coupling shell, which can be used as an interface for multidisciplinary simulations, e.g., for fluid-structure interactions. A very simple quasi one-dimensional test problem is used to show the usage of an preliminary implementation of FLECS, and an overview of the available routines is given. More investigations are needed to prove its functionality, to experiment with different kind of coupling methods, such as nearest neighbor or Gauss interpolation. We would like to extend the parallel capabilities of FLECS to be able to simulate realistic cases on parallel computer platforms.

References

1. A. de Boer, A.H. van Zuijlen and H. Bijl. *Review of coupling methods for non-matching meshes*, Special Issue of Comput. Methods in Appl. Mech. and Eng., accepted for publication, 2006.
2. A. de Boer, A.H. van Zuijlen and H. Bijl. *Coarse grid acceleration of partitioned fluid-structure interaction simulations*, subm. for publ. in Comput. Struct.
3. E.J. Lingen, M. Nool, A. de Boer and H. Bijl. *Design of Flecs, a flexible coupling shell*, 2006, see <http://www.hsa.lr.tudelft.nl/???>
4. William Gropp, Ewing Lusk and Rajeev Thakur. *Using MPI-2 Advanced Features of the Message Passing Interface* The MIT Press, Cambridge, 1999.
5. MpCCI, The Fraunhofer-Institute for Algorithms and Scientific Computing (SCAI). *Multidisciplinary Simulation through Code Coupling*, see <http://www.scai.fraunhofer.de/mpcci.html>
6. A.H. Zuijlen and H. Bijl. *High order time integration schemes for the simulation of non-linear fluid-structure interaction on moving meshes*, in: Proceedings of the Int. Conf. on Comp. Methods for Coupled Problems in Science and Engineering, eds. E. Papadarakakis, E. Oñate, B. Schrefler, Santorini Island, Greece, 25–27 May, 2005.

Flow Solver A	Coupling Server Γ	Structure Solver B
FLECS_Send(Δt^n)	Transfer(Δt^n)	FLECS_Recv(Δt^n)
FLECS_SendDataSet (dset $_{Ap}$, $p_{A,\gamma}^n$)	Receive($p_{A,\gamma}^n$) $\mathbf{p}_{B,\gamma}^n =$ Update($\mathbf{H}_{BA}^n, \mathbf{p}_{A,\gamma}^n$) Send($\mathbf{p}_{B,\gamma}^n$)	FLECS_RecvDataSet (dset $_{Bp}$, $\mathbf{p}_{B,\gamma}^n$)
	$\mathbf{H}_{AB}^n =$ Coupling($\mathbf{x}_{A,\gamma}^n, \mathbf{x}_{B,\gamma}^n$)	Solve structure
FLECS_RecvDataSet (dset $_{Ad}$, $\mathbf{dx}_{A,\gamma}^{n+1}$)	Receive($\mathbf{dx}_{B,\gamma}^{n+1}$) $\mathbf{dx}_{A,\gamma}^{n+1} =$ Update($\mathbf{H}_{AB}^n, \mathbf{dx}_{B,\gamma}^{n+1}$) Send($\mathbf{dx}_{A,\gamma}^{n+1}$)	FLECS_SendDataSet (pset $_B$, $\mathbf{dx}_{B,\gamma}^{n+1}$)
$\mathbf{x}_{A,\gamma}^{n+1} = \mathbf{x}_{A,\gamma}^n + \mathbf{dx}_{A,\gamma}^{n+1}$		
FLECS_SetCoords (pset $_A$, $\mathbf{x}_{A,\gamma}^{n+1}$)	Receive($\mathbf{x}_{B,\gamma}^{n+1}$) Receive($\mathbf{x}_{A,\gamma}^{n+1}$)	FLECS_SetCoords (dset $_{Bd}$, $\mathbf{x}_{B,\gamma}^{n+1}$)
$\mathbf{x}_A^{n+1} = f(\mathbf{x}_A^n, \mathbf{x}_{A,\gamma}^{n+1})$ Solve flow; Compute \mathbf{R}_A^{n+1} finish = $\mathbf{R}_A^{n+1} < \varepsilon$.or. $t^{n+1} > t_e$	$\mathbf{H}_{BA}^{n+1} =$ Coupling($\mathbf{x}_{A,\gamma}^{n+1}, \mathbf{x}_{B,\gamma}^{n+1}$)	
FLECS_Send(finish)	Transfer(finish)	FLECS_Recv(finish)
if finish FLECS_Disconnect FLECS_ShutDown else Compute Δt^{n+1} end	if finish Break both connections end	if finish FLECS_Disconnect FLECS_ShutDown end

Fig. 4. The $n + 1$ -th iteration step of the FSI process expressed in FLECS routines. The superscript n stands for the time step, whereas the subscript A or B indicate the flow solver A or the structure solver B. A vector refers to interface values in case the underscore γ is present. Here, t^n and Δt^n denote the current time and time step, \mathbf{x} and \mathbf{dx} the position and displacement of the coordinates, \mathbf{R}_A^n the residual, ε indicates the required accuracy of \mathbf{x} for FSI iteration. Finally, the boolean **finish** determines whether the program terminates