

USER MANUAL  
version 1.0.0

# MOODY

## Mooring Dynamics

JOHANNES PALM  
CLAES ESKILSSON

Department of Mechanics and Maritime Sciences  
CHALMERS UNIVERSITY OF TECHNOLOGY  
Göteborg, Sweden 2018  
Contact information: [johannes.palm@chalmers.se](mailto:johannes.palm@chalmers.se)



---

# ***CONTENTS***

---

<b>1</b>	<b>Introduction</b>	<b>1</b>
1.1	General description . . . . .	1
1.2	Installation procedure . . . . .	2
1.3	Getting started . . . . .	2
1.4	Debug tip . . . . .	3
<b>2</b>	<b>The input file</b>	<b>5</b>
2.1	Overview . . . . .	5
2.2	General settings . . . . .	6
2.3	Time settings . . . . .	8
2.4	Vertices . . . . .	9
2.5	Cable types . . . . .	10
2.6	Material models . . . . .	12
2.6.1	Common fields . . . . .	12
2.6.2	Available types . . . . .	12
2.7	Cable objects . . . . .	14
2.8	Initial Conditions . . . . .	15
2.8.1	Common fields . . . . .	15
2.8.2	Available types . . . . .	15
2.9	Boundary Conditions . . . . .	17
2.9.1	Common fields . . . . .	17
2.9.2	Available modes . . . . .	18
2.10	Ground models . . . . .	20
2.11	API settings . . . . .	21
2.12	Print settings . . . . .	21
2.13	Numerical settings . . . . .	21
2.14	Static solver . . . . .	21

<b>3</b>	<b>Running the code</b>	<b>23</b>
3.1	User input . . . . .	23
3.1.1	Flags . . . . .	23
3.1.2	Examples . . . . .	24
<b>4</b>	<b>Post processing</b>	<b>25</b>
4.1	Moody post . . . . .	25
4.1.1	Usage . . . . .	25
4.2	MATLAB routines . . . . .	26
4.3	Output file structure . . . . .	27
<b>5</b>	<b>Use Moody as a mooring module</b>	<b>29</b>
5.1	Introduction to the API . . . . .	29
5.2	Interface functions . . . . .	29
5.2.1	Description . . . . .	29
5.3	Input file entries . . . . .	30
5.4	Dynamic mooring restraint in OpenFOAM . . . . .	32
5.5	Fortran and MATLAB interface functions . . . . .	32
5.6	Test the API . . . . .	32
	<b>REFERENCES</b>	<b>33</b>
<b>A</b>	<b>Theory manual</b>	<b>35</b>
A.1	Background . . . . .	35
A.2	Governing equations . . . . .	35
A.2.1	Measures of strain . . . . .	36
A.2.2	Strain rate . . . . .	36
A.2.3	Tension force . . . . .	37
A.3	External forces . . . . .	37
A.3.1	Added mass . . . . .	37
A.3.2	Buoyancy . . . . .	38
A.3.3	Contact . . . . .	38
A.3.4	Drag . . . . .	39
A.4	The Discontinuous Galerkin method . . . . .	39
A.5	Time step schemes . . . . .	40
A.6	Riemann solver . . . . .	41
A.7	<i>hp</i> -adaptivity . . . . .	41
A.7.1	Error indicator . . . . .	42
A.7.2	Shock detection . . . . .	42
A.7.3	Adaptivity control . . . . .	42
A.7.4	Mesh refinement . . . . .	43
A.7.5	<i>p</i> -adaptivity . . . . .	44
A.7.6	<i>h</i> -adaptivity . . . . .	44
A.8	Limiters . . . . .	45
A.9	API boundary conditions . . . . .	46

---

# 1

## *Introduction*

---

### 1.1 GENERAL DESCRIPTION

Moody is developed as a module for computing cable dynamics. It is primarily aimed at solving problems in marine applications. All examples in this manual are therefore different types of mooring configurations for an offshore structure. This manual aims to explain the usage of Moody as a stand-alone solver for mooring dynamics, and as a plug-in module to hydrodynamic codes via the Moody API. When using this version of moody for scientific purposes, please make sure to cite:

1. J. Palm, C. Eskilsson, and L. Bergdahl. An hp-adaptive discontinuous Galerkin method for modelling snap loads in mooring cables. *Ocean Engineering*, 144:266–276, 2017
2. J. Palm. *Mooring Dynamics for Wave Energy Applications*. PhD thesis, Chalmers University of Technology, 2017

Moody comes with a suite of MATLAB<sup>®</sup> scripts used for post-processing. These are optional to use and are simply provided as a way of loading and processing the output data from Moody. The output data format is presented in chapter 4.

In stand-alone mode, Moody is most easily operated through the terminal window. Automated batch runs are typically made through bash-scripting or any other scripting language of choice. Alternatively, a MATLAB interface for running Moody is included in the post processing suite. The simulation is controlled through an input file and standard flag-value pairs e.g. `-f <fileName>`. This mode is used to make stand-alone, uncoupled simulations of cable dynamics. Moody has an automated program interface (API) for coupling to external solvers, e.g. for hydrodynamics. The setup is very similar to using Moody in stand-alone mode with only minor changes to the input file.

The present version of Moody is a discontinuous Galerkin finite element method with high-order Legendre polynomial expansion bases. The formulation is in conservative form, employing a local Lax-Friedrich type Riemann solver for the numerical fluxes between the elements. For more information on the numerical schemes and the equations of motion, the reader is referred to the theory manual.

Moody is a completely stand-alone c++ code with no dependencies apart from the std-lib. It was originally prototyped in Matlab, which is why many input files are labelled '.m' in the tutorials. This is not a requirement of Moody, and any text-file format would serve well as input file for a simulation.

### 1.2 INSTALLATION PROCEDURE

Moody is shipped as a pre-compiled program suite. For installation on a Linux or Apple architecture, simply use a terminal window to run the packed installation script and agree to the licence agreement. Then source the appropriate environment variables for your architecture in moody-1.0.0-Linux/etc/. Ex:

```
./moody-1.0.0-Linux.sh;  
source moody-1.0.0-Linux/etc/bashrcMoody
```

### 1.3 GETTING STARTED

For tests using the API please see the API description in ch. 5. This section simply lists some examples of how to call Moody from the command line, and modify the input file.

To run a 2.5 s simulation on the lindahl125.m input file, and place output one level up, do:

```
cd $moodyDir/tutorials/catenaryValidation;  
moody.x -f lindahl125.m -o ../catenaryResults -time.end 2.5
```

To run tests with two different mesh resolutions on a case, do:

```
moody.x -f inputFile -o N10 -addInput cable1.N 10  
moody.x -f inputFile -o N20 -addInput cable1.N 20
```

To post process a case in Matlab, use readCase to load cable data into a Matlab-structure. View tension propagation as a movie every tenth save timestep through the moodyMovie function.

```
c=readCase('oneDShockOutput');  
moodyMovie('T',c,c.t(1:10:end));
```

## 1.4 DEBUG TIP

If there are any problems running the software, a good starting point is to check the output of the following commands (in a Linux environment):

```
echo $moodyPath  
echo $moodyDir  
which moody.x  
ldd libmoody.so  
echo $LD_LIBRARY_PATH  
echo $PATH
```





---

# 2

## *The input file*

---

### **2.1 OVERVIEW**

An input file is used to setup the mooring system to be studied. An overview of the required input data is given below.

#### **General settings**

Controls environmental aspects like fluid density and gravity. Individual (top-level) keywords.

#### **Time settings**

Controls time step size, start and end time of simulation, etc. Grouped under keyword `time`.

#### **Vertices**

Specifies the location and the number of vertices in the simulation. A top-level array with key `vertexLocations`.

#### **Cable types**

Specify the cross-sectional and material parameters of a mooring cable. Grouped for each type under keyword `cableTypeX`.

#### **Cable objects**

Specify cable properties like end-points, cable type and length. Grouped for each cable under keyword `cableObjectX`.

#### **Boundary conditions**

Specify type and behaviour for all boundary conditions. Grouped for each boundary under keyword `bcX`.

#### **Ground settings**

Specifies ground model type and characteristics. Grouped under keyword `ground`.

### API settings

When used as a mooring module, this contains API-specific information. Grouped under keyword API.

### Print settings

Print settings control the output format of Moody. Grouped under keyword print.

### Numerical settings

Numerical settings control the quadrature order of the simulation. Grouped under keyword numSet.

### Static solver

A way to relax the analytic initial conditions. Grouped under keyword staticSolver.

The input file can have any text-file format, with or without extension. It follows the format of a Matlab script, where each variable name is assigned a value via the = sign, e.g. `time.start = 5` will set the start time of the simulation to 5s. Multiply defined variables are allowed, but be aware that only the first instance of a keyword will be used. Sub-structures and fields of information are assigned through the . sign. The following sections describe each category of information required to setup a Moody simulation.

## 2.2 GENERAL SETTINGS

The general settings control the simulation environment, such as gravitational acceleration and number of dimensions, etc.

### dimensionNumber

The input is 1, 2 or 3 and defines a 1D, 2D or 3D simulation inside Moody. This is a compulsory input. All vector values in the input file must still be set in 3D format. The input order of ALL vector values in the input file is `[x,y,z]`. The values used are then extracted according to:

`dimensionNumber=1`  $\Rightarrow$  1D simulation in `[x]` only

`dimensionNumber=2`  $\Rightarrow$  2D simulation in `[x,z]` only

`dimensionNumber=3`  $\Rightarrow$  3D simulation in `[x,y,z]`

### gravity (1)

Gravity  $\left(9.81[\text{m/s}^2]\right)$  can be toggled on or off by input value 1 or 0.

### waterLevel (inf) [m]

This refers to the still water level z-coordinate. Thus everything is assumed to be submerged by default.

`waterDensity (1000.0) [kg/m3]`

The water density applies to all fluid below the water level.

`airDensity (0.9) [kg/m3]`

The air density applies to all fluid above the water level.

```
dimensionNumber = 2;  
% Optional %  
gravity = 1;  
waterLevel = 0.0; % [m]  
waterDensity = 1000.0; % [kg/m3]  
airDensity = 1.0; % [kg/m3]
```

### 2.3 TIME SETTINGS

The required time settings are described in detail below.

`time.start [s]`

Start time of the simulation.

`time.end [s]`

End time of the simulation.

`time.dt [s]`

Time step size.

`time.scheme (RK3)`

A string specifying the choice of integration method. Choices are:

`eulerFwd`

The time step scheme is based on the simple 1st order Euler forward method.

`RK3`

Third order explicit Runge-Kutta scheme. This is the default.

`cfl`

If present, it tells Moody to adapt the time step size of the simulation to match a given fraction of the CFL condition of the local cable mesh. Typically `cfl=0.9` works well for most applications, but in cases dominated by ground a lower value might be needed for stability. If field `cfl` is present, field `dt` is not used.

```
time.start = 0;
time.end = 10;
time.dt = 1e-4;
time.scheme= 'RK3';
```

## 2.4 VERTICES

Moody's geometry definition is based on vertices that define points in three dimensional space. A vertex is needed to assign a physical location to any cable or boundary condition object. The cell array (Matlab style) `vertexLocations` is used for this purpose. It is important to note that all points must be written in 3D coordinates, see the description of the `dimensionNumber` input in section 2.2. The following example creates vertex 1 and 2 at points `[0 0 -50]` and `[50 0 0]` respectively.

```
vertexLocations = {  
1 [0 0 -50];  
2 [50 0 0]  
};
```

### 2.5 CABLE TYPES

A cable type is used to define the material properties of a cable. Hydrodynamic coefficients, diameter, mass per meter and constitutive relations are all defined here. When defining the actual cable objects, several objects can be of the same cable type. It is analogous to buying a nylon rope, and cutting it into three pieces. The nylon rope properties are contained in the cable type, and the three pieces become three cable objects.

Cable types are defined as structures with a number following the key name `cableType`, and fields specifying the properties. Several cable types can be specified in the same simulation (e.g. `cableType1`, `cableType2`, ...). The required and optional fields of the cable type structure are described below.

`diameter [m]`

The diameter (nominal diameter in the case of chains) of the cable. If not specified, the default value is computed from `gamma0` and `rho`, assuming a constant density in a circular cross-section.

`gamma0 [kg/m]`

The mass per meter of the cable. If not specified, the default value is computed from `diameter` and `rho`, assuming a constant density in a circular cross-section.

`rho [kg/m3]`

The density of the cable material. If not specified, the default value is computed from `diameter` and `rho`, assuming a constant density in a circular cross-section.

`materialModel`

The material model is a substructure of the cable type. It contains information about the constitutive relations of the cable. See section 2.6 for detailed info.

**Optional** (parenthesis indicates default value)

`CDn (0)`

The drag force coefficient in the normal direction of the cable.

`CDt (0)`

The drag force coefficient in the tangential direction of the cable.

`CM or CMn (0)`

The added mass coefficient of the cable in the normal direction of the cable.

`CMt (0)`

The added mass coefficient in the tangential direction of the cable.

```
cableType1.diameter = 0.05;  
cableType1.gamma0 = 1.5; % (default: rho*area)  
cableType1.rho = 7800; % (default: gamma0/area)  
cableType1.materialModel.type = 'bilinearCable';  
cableType1.materialModel.EA = 1e4;  
% Optional %  
cableType1.CDn = 1; % (default: 0)  
cableType1.CDt = 0.1; % (default: 0)  
cableType1.CM = 2; % (default: 0)
```

## 2.6 MATERIAL MODELS

The material model is a substructure of the `cableType` and contains the information about the constitutive relation between axial force and strain in the cable. Moody uses the cable elongation ( $\delta L/L_0$ ) as strain input to the force-strain relation.

### 2.6.1 Common fields

`EA [N]`

The mean axial stiffness of the cable. In nonlinear materials, this value is used as an approximation when computing the initial condition of the cable analytically.

`type`

A string specifying which available type to use. Each type has its own additional input fields, described below.

`xi(0) [Ns]`

The linear internal damping coefficient of the material. The internal damping force is added to the tension force from the material type and is computed as a linear strain-rate dependence as  $T_\xi = \xi \dot{\epsilon}$ .

### 2.6.2 Available types

`type=biLinear`

Computes the axial tension force of the cable according to Hooke's Law, but disallows compressive forces. Thus  $T = \max(EA \epsilon, 0)$ .

`type=exponential`

This type uses an exponential strain-force relation as:

$$T = \max(0, K (e^{a\epsilon} - 1)) . \quad (2.1)$$

Additional fields of input are:

`K [N]`

Basic stiffness. See eq. (2.1).

`a`

The growth-rate of the exponential function. See eq. (2.1).

`type=polynomial`

Implements strain-force relation as polynomial according to

$$T = \max\left(0, \sum_{i=1}^m C_i \epsilon^i\right) , \quad (2.2)$$



where  $C_i$  is the polynomial coefficient of order  $i$  as defined by the **C** field below. Additional fields of input are:

**C** [N]

A vector set of coefficients. First value is the linear coefficient, second is the quadratic strain dependence and so forth. The length of the vector specifies the order of the polynomial strain-force curve. Ex:  
`materialModel.C = [1.0e3 2.0e3 3.0e3 4.0e3];`

### 2.7 CABLE OBJECTS

Cable objects are the main object type in Moody. It specifies a cable of a given unstretched length between two vertices. Naming convention following `cableObject1`, `cableObject2`, etc. is used to define the cable objects in the input file.

`typeNumber`

The cable type number.

`startVertex`

The start vertex number of the cable. This will be used as  $s = 0$ , where  $s$  is the unstretched cable coordinate from 0 to  $L$ .

`endVertex`

The end vertex number of the cable. This will be used as  $s = L$ , where  $s$  is the unstretched cable coordinate from 0 to  $L$ .

`length [m]`

The unstretched length ( $L$ ) of the cable. For some types of initial condition, e.g. `PreStrain`, the field is unused and is overwritten by the computed value.

`IC`

The IC field is a substructure that defines the initial conditions of the cable. There are several options for the type of initial conditions and they are described in section 2.8.

`N`

The number of elements in the cable.

`P`

The polynomial order of the Legendre polynomial basis functions.

```
cableObject1.typeNumber = 1;
cableObject1.startVertex = 1;
cableObject1.endVertex = 2;
cableObject1.length = 100; % [m]
cableObject1.IC.type = 'PreStrain';
cableObject1.IC.eps0 = 0.05; % [-]
cableObject1.N = 10;
cableObject1.P = 7;
```

## 2.8 INITIAL CONDITIONS

The initial conditions are specified individually for each cable object, as a sub-structure of `cableObject`.

### 2.8.1 Common fields

`type`

A string specifying which available type to use. Each type has its own additional input fields, described below.

### 2.8.2 Available types

`type=PreStrain`

The cable is set to a straight line between the two end points. The length of the cable is recomputed based on the distance between the points and the specified initial pre-strain. Gravity is not taken into account in this initial condition.

`eps0`

The initial pre-strain of the cable. Input is dimensionless and can be either a single value or a vector of values. If more than one value is specified, the length of the vector must match the length of `parts`, see below. Each part of the cable will then be given the matching pre-strain.

`parts (1.0)`

The relative part of the cable subject to the strain specified in `eps0`. The input is a vector of the same size as `eps0` and the sum of all the values must be 1. The values of `parts` relates to the unstretched length fractions of the cable.

```
IC.type = 'PreStrain';  
IC.eps0 = [0.01; 0.02];  
IC.parts = [0.5; 0.5];
```

`type=CatenaryStatic`

The cable is set to be an elastic catenary shape at rest. This option only works in 2D and 3D, when gravity is turned on. Ground interaction is detected analytically, so the cable is initially set to lie still exactly at the ground level. No further input is required.

### `type=HalfSine`

Halfsine type is an extension of type `PreStrain`. A sinusoidally varying displacement is imposed on a pre-strained cable. The sinusoidal displacement is in the vertical direction only.

`amplitude (0) [m]`

The amplitude of vertical displacement. When set to 0, the result is the same as for type `'PreStrain'`.

`periods (0.5)`

The number of sinusoidal periods to use along the cable.

`eps0 (0.0)`

The desired pre-strain in the cable before the sinusoidal displacement. The input is a single value that is applied to the whole cable.

```
IC.type = 'HalfSine';  
IC.amplitude = 1; % [m]  
IC.eps0 = 0.01;  
% Optional %  
IC.periods = 0.5;
```

## 2.9 BOUNDARY CONDITIONS

In order for the FEM model to work, the appropriate boundary conditions for the problem must be applied. Boundary conditions are created through the input file in the same `structure.field` way as other objects.

### 2.9.1 Common fields

There are several modes of boundary conditions, requiring different fields of input, but some values are common for all.

`type`

The type describes if the boundary condition is governing the force (Neumann condition) or the position (Dirichlet condition) of the point. The input is either in string format as 'dirichlet' or 'neumann', or a 3 by 1 cell array where the mode of each dimension is specified individually.

`vertexNumber`

Although several objects can be connected to the same vertex, each boundary condition can only be applied to one. The input specifies the vertex number to apply the boundary condition to.

`startTime (startTime) [s]`

When defined, the boundary value is held fixed at that of `bcX.startTime` for  $t < \text{bcX.startTime}$ . The input is either a single value or a 3 by 1 vector specifying the start time of each coordinate direction independently. It has no effect on `mode='fixed'`. The input is in [s] and the default value is the same as the global start time of the simulation.

`endTime (endTime) [s]`

Defines the time at which the boundary condition stops to be active. For  $t > \text{bcX.endTime}$ , the boundary value is held fixed at that of `bcX.endTime`. The input is either a single value or a 3 by 1 vector specifying the end time of each coordinate direction independently. It is not applicable to `mode='fixed'`. The input is in [s] and the default value is the same as the global end time of the simulation.

`rampTime (0) [s]`

Dynamic boundary conditions are ramped up from static start time conditions over a given time interval  $\Delta\tau = \text{bcX.rampTime}$ . The ramp factor  $Q$  is defined as a function of  $\tau = t - \text{startTime}$  as

$$Q = 0.5 \left( \sin \left( \pi \frac{\tau}{\Delta\tau} - \frac{\pi}{2} \right) + 1 \right). \quad (2.3)$$

`dampTime (0) [s]`

Analogous to `rampTime` for smoothly stopping the motion of a boundary condition as the end time is approached.

### `mode`

The input is a string specifying which of the implemented modes that will be used. Each mode has its own additional input fields, both compulsory and optional. They are described below.

#### 2.9.2 Available modes

##### `mode=fixed`

The fixed mode is used to describe a stationary point / constant force.

`value (vertexLocation alt. [0,0,0]) [m alt. N]`

Specify the boundary condition value. If specified for dirichlet degrees of freedom, this value has precedence over information in the vertex location list.

##### `mode=sine`

This mode is used to generate sinusoidally varying values at the boundary. The offset, amplitude, frequency and phase of the sine motion can be specified for each coordinate direction. Scalar valued input is applied to all directions.

`amplitude ([0,0,0]) [m alt. N]`

The amplitude of the sinusoidal excitation.

`frequency ([0,0,0]) [Hz]`

The frequency of oscillation.

`phase ([0,0,0]) [deg.]`

The phase of the excitation.

`centerValue (vertexLocation alt. [0,0,0]) [m alt. N]`

If defined it determines the offset around which the oscillation takes place.

##### `mode=externalPoint`

This mode is used in MoodyAPI. It uses quadratic interpolation to generate intermediate boundary conditions in between coupling times of the external solver. In MoodyAPI, the type must be `dirichlet`, as the coupling is assumed to control the motion of a vertex in Moody, and return a cable force to the external solver. No further input is required in MoodyAPI. For stand-alone Moody, a quadratic interpolation can be set using the following fields, in combination with `bc.startTime` and `bc.endTime`:

`startValue (vertexLocation alt. [0,0,0]) [m alt. N]`

The start value defines the value of the boundary at the beginning of the simulation.

endValue (vertexLocation alt. [0,0,0]) [m alt. N]

The end value defines the value of the boundary at the endTime.

startVelocity ([0,0,0]) [m/s alt. N/s]

The initial rate of change of the boundary at startTime.

mode=externalRigidBody

MoodyAPI comes with a rigid body framework, where the position state of a rigid body is used as input to the API, and where the total mooring forces and moments are returned to the external solver. As in externalPoint, only type=dirichlet is allowed for use with the API.

inputDoFs (6)

The value must be 6 or 7. If 7, the input values are interpreted as a septernion. If 6, input is point position followed by radians of roll, pitch and yaw respectively.

slaves

A list of vertices whose motions are governed by the rigid body.

slavePositions

A list of slave positions relative to the control point position of the body (the vertexNumber of the BC). The order of the points correspond to the order in which the slaves vertices are listed.

```
bc1.type = 'dirichlet';
bc1.mode = 'sine';
bc1.vertexNumber = 2;
bc1.amplitude = [1;0;1]; % [m]
bc1.frequency = [0.5;0;0.5]; % [Hz]
bc1.phase = [90;0;0]; % [deg]
% Optional %
bc1.centerValue = [0;0;0];
bc1.rampTime = 5; % [s]
bc1.startTime = 0;
bc1.endTime = 100;
```

### 2.10 GROUND MODELS

There is at present only one type of ground model implemented in Moody. The `springDampGround` ground model is a combined linear spring and bilinear damper. The contact force acting on each quadrature point of the cable from the ground is a function of the penetration depth and the vertical velocity. Dynamic friction is also included. The input parameters listed below refer to the equations stated in the theory manual of appendix A.

`type`

The `type` field must be set to `'springDampGround'`.

`level`

This sets the level of the ground in the global coordinate system. The ground is assumed to be horizontal and thus have its normal vector aligned with the global  $z$ -axis. Goes into (A.29) as  $z_0$ .

`stiffness [Pa/m]`

The stiffness of the ground. Goes into (A.29) as  $K$ .

`dampingCoeff (1)`

The damping coefficient specifies the fraction of critical damping of the ground. Energy is only dissipated during the penetration of the ground. During the lifting phase, no vertical damping force is applied. The default value is 1, meaning critical damping. Goes into (A.29) as  $\xi$ .

`frictionCoeff (0)`

This is the friction coefficient between any cable and the ground. Goes into (A.30) as  $\mu$ .

`vc (0.001) [m/s]`

The cut off speed for dynamic friction specifies the horizontal speed at which the friction force reaches its full value. This is a numerical relaxation of the friction force for the times when the cable is changing direction. Goes into (A.30) as  $v_\mu$ .

```
ground.type = 'springDampGround';
ground.level = -50; // ground.stiffness = 1e6;
% Optional %
ground.dampingCoeff = 1;
ground.frictionCoeff = 0.3;
ground.vc = 0.1;
```



## 2.11 API SETTINGS

See the API chapter ?? for information about API control parameters in the input file.

## 2.12 PRINT SETTINGS

Print settings control the output folder content and format.

`dt (time.dt) [s]`

The time between each output time in the result directory.

`mode (1)`

The `mode=1` prints all cable values in both modal and nodal space. `mode=0` prints only modal coefficients. Use `moodyPost.x` to extract nodal output, see chapter 4.

`format ('bin')`

Moody prints in binary format by default. `format='txt'` or `format='ascii'` prints in ASCII format. Binary format is significantly faster and more efficient for storage.

## 2.13 NUMERICAL SETTINGS

Numerical settings control the quadrature order of the simulations. The top-level keyword is `numSet`.

`qPointsAdded (2)`

The number of quadrature points more than the polynomial order  $P$ , to use in the simulation. Default is 2, meaning that  $Q = P + 2$  points will be used in each element. Moody uses Gauss-Legendre-Lobatto quadrature which is exact with  $P + 2$  points for the Legendre polynomial of order  $P$ . However, in cases of nonlinear external forces (such as stiff ground interaction) a higher value might be needed to sample the force properly.

## 2.14 STATIC SOLVER

The static solver (`staticSolver`) option enables a number of Euler fwd time steps on. It is still a bit experimental and will probably develop during coming updates. For now, options are:

`relax (0)`

Switch to 1 to enable the dynamic relaxation of the initial condition.

## 2. The input file

---

`relaxFactor (0.9)` Factor to scale the resulting cable velocity after each time step.

`maxIter (100)` How many iterations (of `timeSteps` time steps each) to do. The dynamics of the system is reset after each iteration.

`timeSteps (100)` How many time steps to take during 1 iteration.

---

# 3

## *Running the code*

---

### 3.1 USER INPUT

Moody is operated through the terminal window. Generally, the `-<flagName> <value>` syntax is used to provide run-time information to the computation. All settings apart from the output file name prepends information in the input file.

#### 3.1.1 *Flags*

`-f (moodyInput.m)`

Filename. Must be followed by the filename of the input file.

`-o (input filename without extension)`

Output filename. Must be followed by the name of the output folder.

`-time.start`

Specifies the start time of the simulation. Overrides information specified in input file.

`-time.end`

Specifies the end time of the simulation. Overrides information specified in input file.

`-startState`

This is the reboot flag of moody. If followed by a value, the value should be an existing results folder name. If no value is specified, the default is to use the output folder if it exists. Moody will start from the mooring state at the start time of the simulation. If no results are found, Moody will start from static equilibrium, and print a warning message.

### 3. Running the code

---

#### `-addInput`

A way to specify additional changes to the input file as value pairs: `-addInput <name1> <value1> <name2> <value2>`. All information prepends the input file and thereby overrides the information in the input file.

#### 3.1.2 *Examples*

```
moody.x -f caseFile.m
moody.x -f moodyInput.m -o outFolder -time.start 10
-time.end 20 -addInput time.dt 0.01
```

---

# 4

## *Post processing*

---

### 4.1 MOODY POST

Moody has a post-processing utility named `moodyPost.x`. Although nodal values are printed to the output directory, `moodyPost.x` can be used to create a smaller set of output data for post-processing. It can also be used to generate VTK-files of the cable lines and their tension force magnitude for visualisation in e.g. Kitware's Paraview.

#### 4.1.1 Usage

`moodyPost.x` is a command line tool. The first parameter must be a moody result directory. Default is to process all times in `output/time.dat`. To control which times to process and output there are three flag options.

`-times`

A list of output times.

Ex. `moodyPost.x outName -vtk -times '0,1,2,4.5'`

`-timeList`

Use times from a file. File should be in a single row vector format with no header.

Ex. `moodyPost.x outName -vtk -timeList timeFile.txt`

`-dt [s]`

Use time step size `dt` to step through the output history.

Ex. `moodyPost.x outName -vtk -dt 0.5`

There are further two output options:

**-vtk**

Print VTK files in sub-directory VTK.

Ex. `moodyPost.x moodyResults -vtk -times '0,1,2,4.5'`

**-p**

Print nodal values in sub-directory processed. Optional flag `-var` is used to control which parameters to output. Default is to use all.

`-var` Should be followed by a list of integer values in 1 to 7, where

1. tension
2. strain
3. strain rate
4. position
5. velocity
6. tangent
7. end point values.

Ex. `moodyPost.x moodyResults -p -var '[1,4]'`, prints tension and position.

### 4.2 MATLAB ROUTINES

A set of Matlab routines and functions are provided in `$moodyDir/post/matlab`. These can be used to load and visualise the results. Each cable result is analysed separately. The most important routine is `readCase.m` which returns a cell array of data structures containing the data of all cable objects. The following is an excerpt from the help text of the main functions.

```
>> help readCase
readCase is used to load moody results into an array of data structures.
Input:
  folderName: name of result folder. (default is present dir.)
  varargin : is a list of variables to load. Options are
    'p'   for position  (nq by dim) each time
    'v'   for velocity  (nq by dim) each time
    'tan' for tangent   (nq by dim) each time
    'T'   for tension   (nq by 1) each time
    'eps' for strain    (nq by 1) each time
    'epsR' for strainRate (nq by 1) each time

Output:
  d : output cell array where each cell is a cable structure.
    d.t contains output times
    Each variable field is a nTimes by 1 cell.

>> help probeCable
probeCable extract data at point s or in interval [s0,s1] s\in[0 1]
d is formatted as output from readCase.

If sProbe is single-valued : sProbe is an s-value \in [0, 1].
If not an exact match of output point, a linear interpolation of all
values is made.
```

```

If sProbe = [s0 s1] : sProbe specifies an s-range\in [0,1]. This option
works only for non-adaptive meshes.
out has the fields specified below. It contains only time histories at
the point specified by sProbe.

>> help moody
moody Matlab interface to run moody stand alone, instead of in a terminal.

Input:
  inputName : name of the input file. (including any '.m', '.txt' if applicable)
  varargin : a comma-separated list of (key ,value) input options matching the
            terminal format.
For further information on usage, please see the moody manual.

>> help endPointData
endPointData get values of cable endpoints for cable structure d.
Input:
  d      : cable data structure as returned by readCase.m
  willPlot : switch to plot time history of tension T(:,end)

Output:
  ep      : nTimesteps by 1+2*(3*dim+2) matrix of endpointvalues.
  colHdrs : headers of each column in the output

>> help moodyMovie
moodyMovie shows how parameter "field" evolves over time t.
movie is stored as <d.name>_field.mp4 in the current directory.

Input:
  field: name of field to view.
  d    : cable structure. As output from readCase.m

Optional inputs are:
  t: Vector of plot times. Default is 1 frame per saved timestep
  dims: [i j] (= [1 dim]), i.e. which dimensions to view.
       [0 i] means plotting dimension i against the cable coordinate s.
Varargin input can be:
'fr' : framerate [1/s] , or
'dur' : duration [s]

Example: moodyMovie('v',d,d.t(1:10:end),[0 3], 'dur',20)

Views the time evolution of the z-dimension of cable velocity
along the cable coordinate as a 20 s long movie with frames
every 10th saved timestep of cable structure d.
d is the output of d=readCase('outputName'). See readCase.m

>> help getElementData
elementValues collects the values d at element elNum.
note: this function can only work on nonadaptive meshes.

```

### 4.3 OUTPUT FILE STRUCTURE

The first column of each output file contains the time stamp of the output,  $t = [t_1, t_2, \dots, t_m]$ . This column is also printed to the `time.dat` file in the results directory. Thus all output data files have  $m$  number of rows. The values are then presented along the unstretched cable coordinate  $s$  for each  $n$  output quadrature points  $[s_1, s_2, \dots, s_n]$  of the cable. Output is separated into vector valued output and scalar output. For scalars such as tension, strain and strain rate, the output

**Table 4.1:** Output structure description for cable object data. Times are indexed as  $t_1, t_2, \dots, t_m$ , vector components as  $\bar{f} = [f_x, f_y, f_z]$  and the cable unstretched coordinate  $s$  is indexed as  $s_1, s_2, \dots, s_n$ .

File suffix	Output structure			
_position.dat	$t_1$	$x(t_1, s_1 : s_n)$	$y(t_1, s_1 : s_n)$	$z(t_1, s_1 : s_n)$
	$\vdots$	$\vdots$	$\vdots$	$\vdots$
	$t_m$	$x(t_m, s_1 : s_n)$	$y(t_m, s_1 : s_n)$	$z(t_m, s_1 : s_n)$
_velocity.dat	$t_1$	$v_x(t_1, s_1 : s_n)$	$v_y(t_1, s_1 : s_n)$	$v_z(t_1, s_1 : s_n)$
	$\vdots$	$\vdots$	$\vdots$	$\vdots$
	$t_m$	$v_x(t_m, s_1 : s_n)$	$v_y(t_m, s_1 : s_n)$	$v_z(t_m, s_1 : s_n)$
_tension.dat	$t_1$	$T(t_1, s_1 : s_n)$		
	$\vdots$	$\vdots$		
	$t_m$	$T(t_m, s_1 : s_n)$		
_strain.dat	$t_1$	$\mathcal{E}(t_1, s_1 : s_n)$		
	$\vdots$	$\vdots$		
	$t_m$	$\mathcal{E}(t_m, s_1 : s_n)$		
_sPlot.dat	$t_1$	$s_1 : s_n$		

goes from the start to the end of the cable. For the vector valued output, such as position and velocity, the data is stored consecutively for each coordinate direction of the simulation, in the order  $x, y, z$ . The structure is described in table 4.1.



---

# 5

## *Use Moody as a mooring module*

---

### 5.1 INTRODUCTION TO THE API

Moody is primarily designed to be used as a modular add-on for an external solver for the fluid problem. When Moody is started up in API-mode the external solver is guiding the time evolution of some of Moodys boundary conditions. The interaction between the codes is made as an information loop, see figure 5.1.

When the time step size of the external solver is larger than the internal time step of Moody, a sub-stepping is performed. Moody interpolates in time between the old and the new boundary values received. Quadratic interpolation with optional time staggering is used. See [9] for more information on how different choices of interpolation can affect the solution.

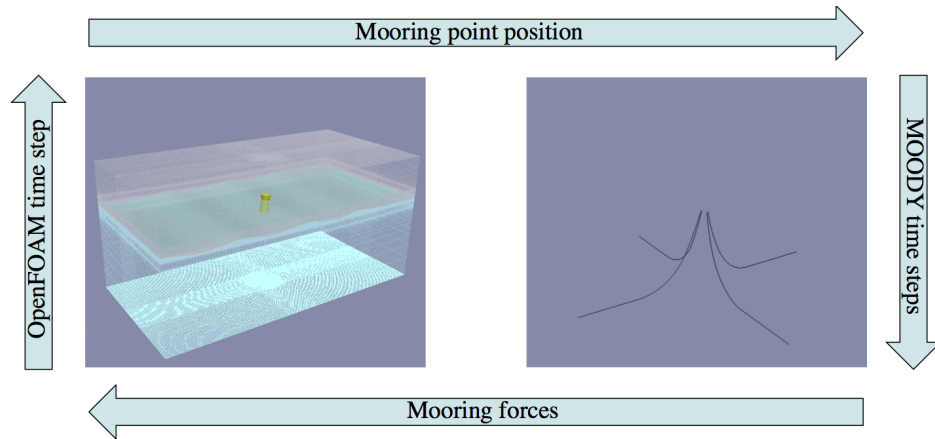
### 5.2 INTERFACE FUNCTIONS

The program interface is made up of three global functions:

- (i) `moodyInit` - an initialisation routine;
- (ii) `moodySolve` - to compute the mooring forces; and
- (iii) `moodyClose` - to close the moody objects and clean the output files.

#### 5.2.1 *Description*

The interface is contained in `$moodyDir/include/moodyWrapper.h`, together with a description of all interface parameters required. The following is an excerpt from `moodyWrapper.h`, showing the definition of the interface functions.



**Figure 5.1:** Schematic description of the information loop between Moody and an external software, represented by an OpenFOAM CFD simulation in this case.

```

/** Initialisation call to moody. Required to setup the API.
Parameter fName:      input file name (including path and extension)
Parameter nVals:      number of values in parameter initialValues
Parameter initialValues: array of initial values of the boundary conditions (
    global frame)
Parameter startTime:   time at start of simulation (s)
*/
void moodyInit(const char* fName, int nVals, double initialValues[], double
    startTime );

/** Solves the internal system of mooring dynamics between time t1 and t2.
Parameter X:      Boundary condition values at time t2.
Parameter F:      Returned as the outward pointing mooring forces for each
    boundary condition dof.
Parameter t1:     Time of last call, start time of time integration.
Parameter t2:     End time of present time step simulation.

Note: t1 and t2 are stored internally, with corresponding states of the
    mooring dynamics. If t1 has increased since the last call, moody will
    move forward in time and store the state at t1 as the new starting state
    of the mooring dynamics. Several iterations can be made with varying t2
    but the same t1 without losing the backup starting state. Compatible
    with predictor/corrector type time stepping schemes.
*/
void moodySolve(const double X[], double F[], double t1, double t2);

/** Closes the moody simulation and stores data for post processing */
void moodyClose();

```

In principle all input files can be used to run an API simulation, but some compulsory items are needed for the coupling to run smoothly.

### 5.3 INPUT FILE ENTRIES

Two modes of boundary condition are supported for control via the API: `externalPoint` and `externalRigidBody`. See section 2.9 for information on their individual

setup. Other input file entries for the API are grouped under the API keyword. The following fields are available to control the simulation behaviour, where some are simply input-file versions of the command-line flags described in chapter 3:

**bcNames**

This is the only compulsory field of the API. It lists the boundary conditions that are to be externally controlled. The order of the input tells Moody how to interpret the input in the `initialValues` parameter of `moodyInit` and in the `X` parameter in `moodySolve`. By extension it also controls the order of the forces returned in parameter `F` in `moodySolve`.

Ex: `API.bcNames = {'bc4'; 'bc5'; 'bc1'};`

**staggerTimeFraction (0.0)**

Used to control the API time staggering for a smoother mooring behaviour. Value  $\alpha$  should be a scalar  $\alpha \in [0, 1]$ , specifying a fraction of the latest time step at which to stop the mooring simulation and return the mooring force. Default is  $\alpha = 0.0$ , but recommended value is  $\alpha = 0.5$ . End time of each coupling step is then computed from  $t_{\text{end}} = (1 - \alpha)t_2 + \alpha t_1$ .

Ex: `API.staggerTimeFraction = 0.5`

**output (input file name)**

The name of the Moody output directory of the simulation. Equivalent to the `-o` flag.

Ex: `API.output = 'moodyResults'`

**reboot (yes)**

This flag controls the startup behaviour of Moody in API mode. Default value is 'yes' and tells Moody to attempt a reboot from a dynamic state at `time.start` in the output directory. New output is then appended to the previous results. To disable use of old results and start from static initial conditions, use `API.reboot='no'`. If `API.reboot` takes any other string value, this should be a result directory that can be used as a source state for the simulation. The source directory is then unchanged by the simulation.

Ex: `API.reboot = 'no'`

**syncOutput (0)**

Integer value 0 or 1 allowed. If 1, Moody results are printed at each coupling time (in addition to at each `saveInterval`). If 0, output is only printed based on Moody's internal `print.dt`.

Ex: `API.syncOutput = 1`

## 5.4 DYNAMIC MOORING RESTRAINT IN OPENFOAM

Again, Moody was built for numerical simulation of the moored motion of wave energy converters (WECs) and other offshore structures. The mooring dynamic restraint on a floating body is an important part of the survivability and reliability of the offshore installation. The original coupling of Moody to an external CFD software was made with OpenFOAM [7] in 2013 [11] using a prototype Matlab version of Moody. This coupling has been updated to compile with the v1712+ version of OpenFOAM. The source code is available in the `thirdParty` folder. The details of required changes to the native code of OpenFOAM are presented in `changesMade.txt` in order to ease migration of the restraint to other versions of OpenFOAM.

In addition to the source code of the mooring restraint and the changes to the motion library, there is in `thirdParty/OpenFOAM` a tutorial (`mooredFloatingObject`) on how to use Moody. It simulates mooring of the classical floating object tutorial of OpenFOAM without the need of any motion constraints.

## 5.5 FORTRAN AND MATLAB INTERFACE FUNCTIONS

There is a Fortran module with interface functions to Moody, to facilitate coupling to Fortran codes. There is also a Matlab API which allows for easy prototyping of coupled simulations from MATLAB. Both interfaces are located in the `thirdParty`.

## 5.6 TEST THE API

Finally, there is also a `testAPI.x` program for testing a Moody api setup. The following example simulates the `mooringSystem.m` run via the time series specified in the `positions.txt` file. The time series should be in simple text-file format with no header line. The first column specifies the time, and the remaining columns specify the values of each api input dof respectively. See the matlab api tutorial for an example of `positions.txt`.

```
testAPI.x mooringSystem.m positions.txt
```

---

## REFERENCES

---

- [1] G. Barter and D. Darmofal. Shock capturing with PDE-based artificial viscosity for DGFEM: Part i. Formulation. *J. Comp. Phys*, 229:1810–1827, 2010.
- [2] P. Bernard. *Discontinuous Galerkin methods for Geophysical Flow Modeling*. PhD thesis, Ecole polytechnique de Louvain, 2008.
- [3] B. Cockburn and C.W. Shu. Runge-Kutta discontinuous Galerkin methods for convection-dominated problems. *J. Sci. Comp.*, 16:173–261, 2001.
- [4] L. Krivodonova, J. Xin, J.-F. Remacle, N. Chevaugeon, and J. Flaherty. Shock detection and limiting with discontinuous Galerkin methods for hyperbolic conservation laws. *J. Appl. Num. Math.*, 48:323–338, 2004.
- [5] J. Lindahl. Implicit numerisk lösning av rörelseekvationerna för en förankringskabel. Technical Report Report Series A:11, Chalmers University of Technology, 1984.
- [6] J.R. Morison, M.P. O’Brien, J.W. Johnson, and S.A. Schaaf. The force exerted by surface waves on piles. *Petroleum Transactions, Amer. Inst. Mining Engineers*, 186:149–154, 1950.
- [7] OpenCFD Ltd. *OpenFOAM Homepage*, 2014. Available <http://www.openfoam.org>.
- [8] Orcina Inc. *OrcaFlex manual – version 9.5a*, 2012.
- [9] J. Palm. *Mooring Dynamics for Wave Energy Applications*. PhD thesis, Chalmers University of Technology, 2017.
- [10] J. Palm, C. Eskilsson, and L. Bergdahl. An hp-adaptive discontinuous Galerkin method for modelling snap loads in mooring cables. *Ocean Engineering*, 144:266–276, 2017.

## REFERENCES

---

- [11] J. Palm, C. Eskilsson, G. Paredes, and L. Bergdahl. CFD simulations of a moored floating wave energy converter. In *Proc. 10th European Wave and Tidal Energy Conference*, Aalborg, Denmark, 2013.

---

# A

## *Theory manual*

---

### A.1 BACKGROUND

The present version of Moody is implemented as an *hp*-adaptive cable solver based on the discontinuous Galerkin method with an approximate Riemann solver of local Lax-Friedrich type. This manual outlines the governing equations and highlights the modelling approach used in the implementation. This manual is mostly compiled from Palm et al [10] where the present formulation was first presented. In addition, the section on API boundary condition interpolation is collected from the PhD thesis of Palm [9].

### A.2 GOVERNING EQUATIONS

For a cable of length  $L_c$ , we use the unstretched cable coordinate  $s \in [0, L_c]$  to express the global coordinate position vector of the cable as  $\mathbf{r} = [r_1(s), r_2(s), r_3(s)]^T$ . Under the assumption of negligible bending stiffness, the equation of motion becomes

$$\gamma_0 \ddot{\mathbf{r}} = \frac{\partial}{\partial s} (T \hat{\mathbf{t}}) + \mathbf{f}, \quad (\text{A.1})$$

$$\hat{\mathbf{t}} = \frac{\partial \mathbf{r}}{\partial s} \left| \frac{\partial \mathbf{r}}{\partial s} \right|^{-1}, \quad (\text{A.2})$$

where  $\gamma_0$  is the cable mass per unit length,  $T$  is the cable tension force magnitude,  $\hat{\mathbf{t}}$  is the tangential unit vector of the cable and  $\mathbf{f}$  represents all external forces. For notation we use  $\dot{x} = \frac{\partial x}{\partial t}$  to indicate time derivatives and  $|\mathbf{x}| = \sqrt{x_i x_i}$  to denote the  $L_2$  - norm of a vector quantity  $x$ , Vector components are denoted by their index as  $x_i$ ,  $i \in [1, 2, 3]$ , and summation over repeated indices is implied.

Written as a first order system in terms of the cable position  $\mathbf{r}$ , its spatial derivative  $\mathbf{q} = \frac{\partial \mathbf{r}}{\partial s}$  and its momentum density  $\mathbf{v} = \dot{\mathbf{r}}\gamma_0$ , eq. (A.1) becomes

$$\dot{\mathbf{r}} = \frac{\mathbf{v}}{\gamma_0}, \quad (\text{A.3})$$

$$\dot{\mathbf{q}} = \frac{\partial}{\partial s} \left( \frac{\mathbf{v}}{\gamma_0} \right), \quad (\text{A.4})$$

$$\dot{\mathbf{v}} = \frac{\partial}{\partial s} (T\hat{\mathbf{t}}) + \mathbf{f}, \quad (\text{A.5})$$

where we have assumed that the cable mass is constant in time. In terms of a state vector  $\mathbf{u} = [\mathbf{r}, \mathbf{q}, \mathbf{v}]^T$  the conservative form of the problem is written as

$$\dot{\mathbf{u}} = \frac{\partial \mathbf{F}(\mathbf{u})}{\partial s} + \mathbf{Q}(\mathbf{u}), \quad (\text{A.6})$$

with a flux function

$$\mathbf{F}(\mathbf{u}) = \left[ \emptyset, \frac{\mathbf{v}}{\gamma_0}, T\hat{\mathbf{t}} \right]^T, \quad (\text{A.7})$$

and a non-linear source term

$$\mathbf{Q}(\mathbf{u}) = \left[ \frac{\mathbf{v}}{\gamma_0}, \emptyset, \mathbf{f} \right]^T. \quad (\text{A.8})$$

### A.2.1 Measures of strain

Constitutive modelling in Moody is based on a strain-tension relation. We use the elongation of the cable as a measure for the strain throughout the code. This is to be most easily compatible with nonlinear strain-tension curves that originate from physical experiments on cables. Therefore, it is assumed that input parameters to material models are in accordance with this criteria. The elongation strain  $\varepsilon$  is computed from

$$\varepsilon = \sqrt{\mathbf{q} \cdot \mathbf{q}} - 1, \quad (\text{A.9})$$

so that

$$|\mathbf{q}| = 1 + \varepsilon, \quad (\text{A.10})$$

$$\hat{\mathbf{t}} = \frac{\mathbf{q}}{1 + \varepsilon} \quad (\text{A.11})$$

### A.2.2 Strain rate

The strain rate is not formally included in the formulation. It is computed from

$$\dot{\varepsilon}^{(k)} = \dot{\mathbf{q}}^{(k-1)} \cdot \hat{\mathbf{t}}^{(k)}, \quad (\text{A.12})$$



where superscript  $(k)$  indicates at which time step the variable is evaluated.  $\dot{\mathbf{q}}^{(k-1)}$  thus means that the value from the last time step is used to approximate the strain rate. For highly damped cables, a lower CFL condition might be needed to ensure stability due to this approximation.

### A.2.3 Tension force

The axial tension force is evaluated as  $T = T(\varepsilon, \dot{\varepsilon})$ , for each material model implemented. As an example, the `biLinear` cable model has a truly linear relation between  $T$  and  $\varepsilon$ , with the stiffness  $EA$ . There is no deviation from the linear curve due to cross-sectional area diminishing for large strains, as that is assumed to be inherent in the input relation between elongation and tension. Cross-sectional area change due to strain is taken into account in the evaluation of the viscous drag force on the cable, see section A.3.4.

## A.3 EXTERNAL FORCES

The total external force  $\mathbf{f}$  of eq. (A.1) is given by

$$\mathbf{f} = \mathbf{f}_a + \mathbf{f}_b + \mathbf{f}_c + \mathbf{f}_d, \quad (\text{A.13})$$

where  $\mathbf{f}_a$  is the added mass and Froude-Krylov forces,  $\mathbf{f}_b$  is the net force of gravity and buoyancy,  $\mathbf{f}_c$  represent contact forces, typically from sea-floor interaction, and  $\mathbf{f}_d$  is the viscous drag force.

For notation, a vector quantity  $\mathbf{x}$  can be expressed in a tangential and normal component with respect to the cable tangent respectively as

$$\mathbf{x}_{\hat{t}} = \mathbf{x} \cdot \hat{\mathbf{t}}, \quad (\text{A.14})$$

$$\mathbf{x}_{\hat{n}} = \mathbf{x} - \mathbf{x}_{\hat{t}}. \quad (\text{A.15})$$

### A.3.1 Added mass

The added mass effect on the cable dynamics is divided into a fluid part  $\mathbf{f}_{a+}$  dependent on the fluid acceleration, and an inertial part  $\mathbf{f}_{a-}$  which scales with the cable acceleration,

$$\mathbf{f}_a = \mathbf{f}_{a+} - \mathbf{f}_{a-}. \quad (\text{A.16})$$

The added mass contributions are computed from Morison's equations [6] using coefficients  $C_{Mn}$  and  $C_{Mt}$  in the normal and tangential direction respectively.

$$\mathbf{f}_{a+} = A_c \rho_f \left( (1 + C_{Mn}) \dot{\mathbf{v}}_f + (C_{Mt} - C_{Mn}) \dot{\mathbf{v}}_{f\hat{t}} \right), \quad (\text{A.17})$$

$$\mathbf{f}_{a-} = \frac{A_c \rho_f}{\gamma_0} (C_{Mn} \dot{\mathbf{v}} + (C_{Mt} - C_{Mn}) \dot{\mathbf{v}}_{\hat{t}}), \quad (\text{A.18})$$

with  $\rho_f$  as the fluid density and  $\mathbf{v}_f$  as the fluid velocity.

The inertial contribution is moved to the left hand side of eq. (A.5), effectively changing it to

$$\mathbf{F}_{\text{tot}} = \frac{\partial}{\partial s} (T\hat{\mathbf{t}}) + \mathbf{f} + \mathbf{f}_{a-}, \quad (\text{A.19})$$

$$\mathbf{F}_{\text{tot}} = \dot{\mathbf{v}} + \mathbf{f}_{a-}. \quad (\text{A.20})$$

The computation of  $\dot{\mathbf{v}}$  is then reduced to solving

$$\mathbf{F}_{\text{tot}} = [a\mathbf{I} + b\hat{\mathbf{t}} \otimes \hat{\mathbf{t}}] \dot{\mathbf{v}} = \boldsymbol{\psi} \dot{\mathbf{v}}, \quad (\text{A.21})$$

$$a = 1 + A_c \rho_f \frac{C_{\text{Mn}}}{\gamma_0}, \quad (\text{A.22})$$

$$b = A_c \rho_f \frac{C_{\text{Mt}} - C_{\text{Mn}}}{\gamma_0}, \quad (\text{A.23})$$

where  $\mathbf{I}$  is the  $3 \times 3$  identity matrix,  $\boldsymbol{\psi}$  is the added inertia matrix and  $\otimes$  denotes the open vector product. In effect, this is then handled as if  $\boldsymbol{\psi}^{-1}$  was a relaxation step on  $\dot{\mathbf{v}}$  for each computational node:

$$\dot{\mathbf{v}} = \boldsymbol{\psi}^{-1} \mathbf{F}_{\text{tot}}, \quad (\text{A.24})$$

and the inverse matrices  $\boldsymbol{\psi}^{-1}$  are analytically obtained for each node as

$$\boldsymbol{\psi}^{-1} = \frac{1}{a(a+b)} ((a+b)\mathbf{I} - b\hat{\mathbf{t}} \otimes \hat{\mathbf{t}}). \quad (\text{A.25})$$

### A.3.2 Buoyancy

The buoyancy force is straightforward and computed from gravitational acceleration  $\mathbf{g}$ , the cable density  $\rho_c$ , the fluid density  $\rho_f$ , and the cable mass  $\gamma_0$  as

$$\mathbf{f}_b = \mathbf{g} \frac{\rho_c - \rho_f}{\rho_c} \gamma_0. \quad (\text{A.26})$$

$$(\text{A.27})$$

### A.3.3 Contact

The implementation of the ground model has a potentially large influence on the quality of the simulation. Moody supports contact forces from the ground based on a bilinear spring and damper. The implementation is close to that used in [8]. A tangential friction model from [5] using dynamic friction is also implemented. For a horizontal sea floor with  $(x, y, z)$  coordinates corresponding to vector index

1, 2 and 3 respectively, the contact force is computed as

$$\mathbf{f}_c = \begin{cases} \mathbf{G}_v + \mathbf{G}_h & \text{if } (z_G - r_{\hat{z}}) \geq 0, \\ 0 & \text{otherwise,} \end{cases} \quad (\text{A.28})$$

$$\mathbf{G}_v = \left( K_G d_c (z_G - r_{\hat{z}}) - 2\xi_G \sqrt{K_G \gamma_0 d_c} \min(\dot{r}_{\hat{z}}, 0) \right) \hat{\mathbf{z}}, \quad (\text{A.29})$$

$$\mathbf{G}_h = \mu f_{b\hat{z}} \tanh\left(\frac{\pi \dot{r}_{\hat{xy}}}{v_\mu}\right) \frac{\dot{\mathbf{r}} - \dot{r}_{\hat{z}} \hat{\mathbf{z}}}{|\dot{r}_{\hat{xy}}|} \quad (\text{A.30})$$

where indices  $\hat{xy}$ , and  $\hat{z}$  denote the horizontal and vertical projections of a vector respectively. Further,  $r_{\hat{z}}$  is the vertical coordinate of the cable position,  $z_G$  is the vertical position of the ground,  $K_G$  is the ground stiffness,  $\xi_G$  is the ratio of critical damping for the ground–cable pair. The coefficient of kinetic friction is in eq (A.30) denoted as  $\mu$ , with a corresponding velocity of maximum friction  $v_\mu$ .

Please note that to model the ground interaction as described in eq. (A.28), we need the position of the cable  $r$  as an independent variable, which is why it is included as an independent variable in eq. (A.3).

#### A.3.4 Drag

The viscous drag contribution is modelled via Morison’s formulas [6] for a circular cross-section with volume-preserving properties during axial strain. Hence, the elongation factor  $(1 + \varepsilon)$  and the contraction factor  $(1 + \varepsilon)$  do not fully cancel. This is why the contraction factor for the diameter is  $(\sqrt{1 + \varepsilon})^{-1}$ , and  $\mathbf{f}_d$  is expressed as

$$\mathbf{f}_D = 0.5 \rho_f d_c \sqrt{1 + \varepsilon} (C_{Dt} |\mathbf{v}_t^*| \mathbf{v}_t^* + C_{Dn} |\mathbf{v}_n^*| \mathbf{v}_n^*), \quad (\text{A.31})$$

where  $C_{Dn}$ ,  $C_{Dt}$  are the drag coefficients of the normal and tangential direction, and  $\mathbf{v}^*$  is the relative velocity between fluid and the cable:

$$\mathbf{v}^* = \mathbf{v}_f - \dot{\mathbf{r}}. \quad (\text{A.32})$$

## A.4 THE DISCONTINUOUS GALERKIN METHOD

The discontinuous Galerkin method is used for spatial discretisation of the cable domain. Consider eq. (A.6) for a cable domain  $\Omega$  of unstretched coordinate  $s \in [0, L_c]$ .  $\Omega$  is discretised in a set of  $N$  elemental regions  $\Omega^e$  of mesh size  $h^e$ . In each element a function  $y(s, t)$  is approximated as a Legendre polynomial of order  $p$  as

$$y(s, t) \approx y^e(s, t) = \sum_{k=0}^{k=p} \phi_k(s) \tilde{y}_k^e(t), \quad (\text{A.33})$$

where  $\tilde{y}_k^e$  is the  $k^{th}$  expansion coefficient corresponding to the basis function of order  $k$ ,  $\phi_k(s)$ . The discontinuous Galerkin (DG) formulation of eq. (A.6) is then

$$\left(\phi_k, \frac{\partial u^e}{\partial t}\right)_{\Omega^e} = \left(\phi_k, \frac{\partial F^e}{\partial s}\right)_{\Omega^e} + (\phi_k, Q^e)_{\Omega^e}, \quad \forall k \in [0, p], \quad (\text{A.34})$$

using

$$(a(s, t), b(s, t))_{\Omega^e} = \int_{\Omega^e} a(s, t) b(s, t) ds, \quad (\text{A.35})$$

to denote the inner product operator on the elemental domain. The Gauss-Lobatto-Legendre quadrature points and quadrature rules are used for numerical evaluation of the elemental integrals in (A.35)

Two integration by parts of the weak derivative term  $\left(\phi_k, \frac{\partial F^e}{\partial s}\right)_{\Omega^e}$ , allows eq. (A.34) to be written

$$\left(\phi_k, \frac{\partial u^e}{\partial t}\right)_{\Omega^e} = \left(\phi_k, \frac{\partial F^e}{\partial s}\right)_{\Omega^e} + \left[\phi_k \left(\widehat{F^e} - F^e\right)\right]_{s_L^e}^{s_U^e} + (\phi_k, Q^e)_{\Omega^e}, \quad (\text{A.36})$$

$\forall k \in [0, p]$ . Here  $s_U^e$  and  $s_L^e$  denote the elemental upper and lower bounds of the unstretched cable domain coordinate  $s$  respectively. Please note the introduction of the numerical boundary flux  $\widehat{F^e}$ . As the elements are discontinuous at the element boundaries, the numerical flux provides the coupling between neighbouring elements.

Finally, separation of time and space dependence defined in eq. (A.33) allows us to rewrite eq. (A.36) in terms of the modal coefficients of our polynomial space:

$$(\phi_k, \phi_i) \frac{\partial \tilde{u}^e}{\partial t} = \left(\phi_k, \frac{\partial \phi_i}{\partial s}\right) \tilde{F}^e + \left[\phi_k \left(\widehat{F^e} - F^e\right)\right]_{s_L^e}^{s_U^e} + (\phi_k, Q^e)_{\Omega^e} \quad (\text{A.37})$$

$\forall k, i \in [0, p]$ .

## A.5 TIME STEP SCHEMES

We use the strong stability preserving 3rd order Runge-Kutta (RK) scheme as described in [3] to advance eq. (A.6) in time. Let  $\mathbf{u}_k^{(i)} = \mathcal{L}(\mathbf{u}_k^{(i)})$ , where  $\mathbf{u}_k^{(i)}$  is the state vector at time step  $k$  and sub-cycle  $i$ . The 3rd order RK scheme is then

$$\begin{aligned}
\mathbf{u}_{k+1}^{(0)} &= \mathbf{u}_k, \\
\mathbf{u}_{k+1}^{(1)} &= \mathbf{u}_{k+1}^{(0)} + \Delta \mathcal{L} \left( \mathbf{u}_{k+1}^{(0)} \right), \\
\mathbf{u}_{k+1}^{(2)} &= \frac{1}{4} \left( \mathbf{u}_{k+1}^{(1)} + \Delta \mathcal{L} \left( \mathbf{u}_{k+1}^{(1)} \right) \right) + \frac{3}{4} \mathbf{u}_{k+1}^{(0)}, \\
\mathbf{u}_{k+1}^{(3)} &= \frac{2}{3} \left( \mathbf{u}_{k+1}^{(2)} + \Delta \mathcal{L} \left( \mathbf{u}_{k+1}^{(2)} \right) \right) + \frac{1}{3} \mathbf{u}_{k+1}^{(0)}, \\
\mathbf{u}_{k+1} &= \mathbf{u}_{k+1}^{(3)}.
\end{aligned} \tag{A.38}$$

## A.6 RIEMANN SOLVER

The numerical fluxes are central to the performance of the numerical scheme used in Moody. The present version uses a local Lax-Friedrich type flux, with  $\widehat{F}^e$  in (A.37) evaluated as

$$\widehat{F}^e = \{F^e\} + |\lambda|_{\max} [[u^e]] \quad \text{on } \Gamma \in \Omega. \tag{A.39}$$

Notations  $\{x^e\}$  and  $[[x^e]]$  are

$$\{x^e\} = 0.5 (x^+ + x^-), \tag{A.40}$$

$$[[x^e]] = 0.5 (n^+ x^- + n^- x^+), \tag{A.41}$$

where  $x^+$  means taking the value from the internal side of the boundary and  $x^-$  from the neighbouring element. The normal vector  $n$  refers to the outward pointing normal, defined for each element boundary as  $n^+ = -1$  on  $s_L^e$  and as  $n^+ = 1$  on  $s_R^e$ . The maximum eigenvalue needed in eq. (A.39) is in this version chosen for each equation as:

$$|\lambda|_{\max} = \begin{cases} 0.8 \left| \frac{[[F^e]]}{[[u^e]]} \right| & \text{if } u = q \\ |c_t| & \text{otherwise} \end{cases}, \tag{A.42}$$

where  $c_t$  is the tangential wave speed in the cable. It is defined by the maximum eigenvalue of the hyperbolic problem,

$$c_t = \sqrt{\frac{\partial T}{\partial \varepsilon}} \mathcal{N}_0. \tag{A.43}$$

## A.7 *hp*-ADAPTIVITY

The goal of the adaptive mesh refinement scheme is to limit the discretisation error to below a pre-set tolerance level,  $\varepsilon^*$ . To do this we need an indicator of the relative error, a smoothness indicator to let us know which type of error that we observe, and a mechanism to efficiently adapt the spatial discretisation to best fit the solution.

### A.7.1 Error indicator

In this work, we use the tension force magnitude  $T$  as indicator variable for the quality of the solution. For smooth solutions, we expect a convergence rate of  $\mathcal{O}(h^{p+1})$ , but close to discontinuities the solution converges as  $\mathcal{O}(h)$  [4]. The elemental jump of the solution is a common measure of the numerical error [4, 2, 1]. Barter et al. [1] used the relative jump

$$\tau = \left| \frac{[[T]]}{\{T\}} \right|,$$

at element edges to indicate regions of shocks. We incorporate the relative jump into Bernard's formula for the relative error [2] to get

$$\varepsilon^e = \frac{1}{\sqrt{8}} \sqrt{(\tau_L^e)^2 + (\tau_R^e)^2}, \quad (\text{A.44})$$

with  $\tau_L^e$  and  $\tau_R^e$  as the relative jumps at the left and right elemental borders respectively.

### A.7.2 Shock detection

We follow Krivodonova [4] in order to decide on the nature of the error, i.e. locate regions of sharp gradients or shocks. An intermediate convergence rate  $h^{0.5(p+1)}$  is used to establish a shock indicator of element  $e$ . Computing

$$I^e = \max(\tau_L^e, \tau_R^e) h^{-0.5*(p+1)}, \quad (\text{A.45})$$

results in the shock criteria as  $I^e \geq 1$  [4]. When the cable goes slack ( $T \rightarrow 0$ ), a snap load is expected to occur in the near future. We therefore treat it as a shock criteria. Thus, we also introduce a low-tension criteria  $T^*$ , that indicate shock behaviour if  $\min(T^e) \leq T^*$ , with  $\min(T^e)$  evaluated at the quadrature points of element  $e$ . The shock criteria and detectors are combined into the shock detecting function  $S^e$  as

$$S^e = \begin{cases} 1 & \text{if } I^e \geq 1 \text{ and } \varepsilon^e \geq \varepsilon^* \\ 1 & \text{if } \min(T^e) \leq T^* \\ 0 & \text{otherwise} \end{cases}. \quad (\text{A.46})$$

Please note that the jump-based shock detector is only applied to elements that have errors higher than the tolerance level, while the low tension indicator is applied to all elements in the cable domain.

### A.7.3 Adaptivity control

We let  $p$ -refinement have precedence over  $h$ -refinement in all elements of smooth solution. If shocks are detected, the mesh is forced into maximum  $h$ -refinement

and the order is reduced to linear approximation  $p = 1$ .  $h$ -refinement is restricted to splitting elements in half and merging two equally sized elements with the same parent. The initial mesh  $h$ -resolution is not allowed to coarsen, and the splitting hierarchy of elements is therefore confined to one element of the initial mesh.

The inverse operation of coarsening the resolution follows the recipe of Bernard [2], assuming the error relation between the new and the old mesh to be:

$$\frac{\varepsilon^e}{\varepsilon^*} = \frac{h^{p+1}}{h^{*p+1}}, \quad (\text{A.47})$$

where  $p^*$  and  $h^*$  denote the resolution parameters in the modified mesh. Thus, for merging elements with constant order  $p$ , the criterion is:  $\varepsilon^e < 0.5^{p+1}\varepsilon^*$ ; and for lowering the polynomial order under constant  $h$ , consequently:  $\varepsilon^e < h\varepsilon^*$ . Combining the above, we arrive at the  $hp$ -adaptive control algorithm

$$\begin{aligned} &\text{if } S^e = 1 : \quad p = 1, \quad h = h_{\min} \\ &\text{if } S^e = 0 : \quad \begin{cases} p = \min(p+1, p_{\max}) & \text{if } \varepsilon^e \geq \varepsilon^* \\ h = \max(0.5h, h_{\min}) & \text{if } \varepsilon^e \geq \varepsilon^* \text{ and } p = p_{\max} \\ h = \min(2h, h_{\max}) & \text{if } \varepsilon^e < 0.5^{p+1}\varepsilon^* \\ p = \max(p-1, 1) & \text{if } \varepsilon^e < h\varepsilon^* \text{ and } h = h_{\max} \end{cases}, \end{aligned}$$

where  $S^e$  is the shock indicator function explained in eq. (A.46). The application of the control algorithm to pure  $h$  is straightforward, as we simply skip the  $p$ -adaptation parts. For pure  $p$  refinement, we skip the  $h$ -adaptation as well as the shock detection step. The adaptive scheme is applied after a complete time step, so that all stages of the Runge-Kutta scheme share the same spatial resolution.

#### A.7.4 Mesh refinement

The  $hp$ -adaptive mesh refinement in Moody is implemented in the following way. First, let  $h$  be the non-dimensional element size,  $P$  be the polynomial order of an element, and  $H$  denote the global level of  $h$ -refinement. For a given target change in mesh with  $\partial H$  levels and  $\partial P$  orders, the mesh is changed according to

(i)  $p$ -refinement,

$$P = P + \max(\partial H, 0); \quad (\text{A.48})$$

(ii)  $h$ -adaptivity, including splitting and merging of elements,

$$H = H + \partial H; \quad (\text{A.49})$$

(iii)  $p$ -coarsening,

$$P = P + \min(\partial H, 0), \quad (\text{A.50})$$

to avoid loss of accuracy.

### A.7.5 *p*-adaptivity

We use the Legendre polynomials as hierarchic modal basis functions. Increasing the polynomial order of an element is therefore reduced to simply padding the solution vector  $\tilde{u}_i$  with 0 when  $P$  is increased. Conversely, *p*-coarsening is achieved by truncating  $\tilde{u}_i$  at the target order.

### A.7.6 *h*-adaptivity

In *h*-adaptivity, elements are added or removed dynamically as the solution progresses. Each element has two level flags used in to keep track of the *h*-refinement hierarchy. A global level index  $I_G$  and a local level index  $I_L$ , controlled during splitting and merging according to:

$$e \rightarrow (e^-, e^+) \begin{cases} I_G^- = I_G^+ = I_G + 1 \\ I_L^- = \min(I_L, 0) - 1 \\ I_L^+ = \max(I_L, 0) + 1 \end{cases}, \quad (\text{A.51})$$

$$(e^-, e^+) \rightarrow e \begin{cases} I_G = I_G^+ + 1 = I_G^- + 1 \\ I_L = I_L^- + I_L^+ \end{cases}. \quad (\text{A.52})$$

Here we use notations  $-$  and  $+$  to denote the left and right child element respectively. The prerequisites for allowing merging of two neighbouring elements who both have been flagged for *h*-coarsening can now be simplified to

$$I_L^- < 0, \quad \text{and} \quad I_L^+ > 0.$$

The global order flag is only used to avoid refinement beyond a pre-set maximum limit and to keep track of the elemental size.

When an element is marked for *h*-refinement, it is divided in two equally sized elements. Thus the local Gauss-Lobatto-Legendre (GLL) quadrature points,  $\xi \in [-1, 1]$ , of the original parent element will always have the same relation to the points of the two new child elements. This allows for a constant, pre-computed split map matrix  $S_s$  between the modal values of the parent and its children. If superscript  $-$  and  $+$  describe values in the left and right child elements respectively, then

$$\begin{bmatrix} \tilde{u}^- \\ \tilde{u}^+ \end{bmatrix} = S_s \tilde{u}, \quad (\text{A.53})$$

Let  $S_{m2n}$ , and  $S_{n2m}$  denote the mapping matrices between the modal solution and the nodal values at the GLL points  $u$  of an element, i.e.

$$\begin{aligned} u &= S_{m2n} \tilde{u} = \phi(\xi) \tilde{u}, \\ \tilde{u} &= S_{n2m} u = M^{-1} W u, \end{aligned}$$



where  $M^{-1}$  is the inverse mass matrix and  $W$  is the quadrature weighted  $\phi^T(\xi)$  matrix. Here we use the matrix  $\phi(\xi)$  as a  $q$  by  $p+1$  matrix where  $\phi_{ij}$  contains the value of basis  $j$  at quadrature point  $i$ . Then the splitting transform  $S_s$  is constructed from

$$S_s = \begin{bmatrix} S_{n2m} & \emptyset \\ \emptyset & S_{n2m} \end{bmatrix} \phi(\xi_s), \quad (\text{A.54})$$

$$\xi_s = \begin{bmatrix} 0.5(\xi - 1) \\ 0.5(\xi + 1) \end{bmatrix}, \quad (\text{A.55})$$

where  $\xi_s$  are the GLL points of the split elements projected on the local parent domain.

The merging operation is the inverse of the splitting, but is computed as a separate transform  $S_m$ . To avoid the ambiguous midpoint of the sibling elements, the merge-matrix is based on an even number of quadrature points in the final (parent) element. If  $Q$  is odd,  $Q+1$  is used to compute the merge matrix.

So, assuming that  $Q$  is even,  $S_m$  is constructed from

$$S_m = S_{n2m}(Q) \begin{bmatrix} \phi(2\xi + 1), \forall \xi < 0 \\ \phi(2\xi - 1), \forall \xi > 0 \end{bmatrix}. \quad (\text{A.56})$$

## A.8 LIMITERS

The generalised minMod slope limiter as it is described in [3] is used to avoid overshoots in the presence of shocks. It is not strictly monotone and TVD as the original minMod method, but is classified as a TVB (Total Variation Bounded) method.

$$\tilde{u}_1^e = \text{minMod} \left( \theta_l \frac{\tilde{u}_0^e - \tilde{u}_0^{e-1}}{2}, \theta_l \frac{\tilde{u}_0^{e+1} - \tilde{u}_0^e}{2}, \tilde{u}_1^e \right), \theta_l \in [1, 2], \quad (\text{A.57})$$

where  $\tilde{u}_0^e$  is the mean value and  $\tilde{u}_1^e$  is the linear slope of element  $e$ . The min-mod function is defined as

$$\text{minMod}(a, b, c) = \begin{cases} \min(a, b, c) & \text{if } a, b, c > 0 \\ \max(a, b, c) & \text{if } a, b, c < 0 \\ 0 & \text{otherwise} \end{cases}.$$

The value of  $\theta_l$  blends the limiter between the classical min-mod for  $\theta_l = 1$  and the less restrictive, generalised min-mod from [3] for  $\theta_l = 2$ . The limiter is applied after each stage of the Runge-Kutta time-stepping scheme.

## A.9 API BOUNDARY CONDITIONS

When coupled to an external solver (i.e. in API mode), the timestep requirement on the cable dynamics is potentially orders of magnitude smaller than that required by the external solver. To save computational effort, the boundary conditions that are sent to Moody at each coupling time are interpolated to achieve intermediate boundary conditions. The position of the attachment point is interpolated based on constant acceleration over the time step, and using a staggered process to maintain smoothness.

To explain, we let  $t_k$  and  $t_{k+1}$  be two consecutive coupling times, with corresponding mooring point positions  $r_k$  and  $r_{k+1}$ . We introduce the staggered-time fraction  $\phi \in [0, 1]$  and identify a corresponding mooring time  $t_{k+1}^m \in [t_k, t_{k+1}]$  as

$$t_{k+1}^m = \phi t_k + (1 - \phi) t_{k+1}. \quad (\text{A.58})$$

The mooring boundary conditions  $r_D(t)$  and  $v_D(t)$  are interpolated over the mooring time step interval  $t \in [t_k^m, t_{k+1}^m]$  as

$$r_D(t_k^m + \tau) = r_k^m + v_k^m \tau + 0.5 a_k \tau^2, \quad (\text{A.59})$$

$$v_D(t_k^m + \tau) = v_k^m + a_k \tau, \quad (\text{A.60})$$

where  $\tau \in [0, t_{k+1}^m - t_k^m]$ , while  $r_k^m = r_D(t_k^m)$  and  $v_k^m = v_D(t_k^m)$  are taken from the previous coupling interval. To close the system, we only need to define  $a_k$ . Here, we choose  $a_k$  as the constant acceleration needed to satisfy  $r_D(t_{k+1}) = r_{k+1}$ , i.e.

$$a_k = \frac{r_{k+1} - r_k^m - v_k^m \Delta_k}{0.5 \Delta_k^2}, \quad (\text{A.61})$$

with  $\Delta_k = t_{k+1} - t_k^m$ .