XML GUIDE FOR DUALSPHYSICS

FLEXIBLE FLUID-STRUCTURE INTERACTION

SPECIAL: FLEXSTRUCS

October 2022

DualSPHysics team

http://dual.sphysics.org

flexstrucs: Flexible fluid-structure interaction

A **flexible fluid-structure interaction** formulation is available in DualSPHysics v5.2. The approach is based on a Total Lagrangian SPH formulation for the dynamics of the flexible structure, and the fluid-structure interaction is handled via the existing dynamic boundary condition within DualSPHysics.

There are three main developments implemented within the structural dynamics solver: 1) a **Total Lagrangian formulation**, which relies on representing all relevant quantities and operators with respect to an initial (material) reference frame, **to eliminate tensile instabilities**; 2) a **kernel correction** scheme for the flexible structures **to retain linear consistency** at the edges of the geometry; 3) an **hourglass (zero-energy) mode suppression** scheme to **reduce the effect of instabilities related to rank-deficiency** of the collocated SPH discretisation.

The **interaction at the fluid-structure interface** is handled automatically by the existing **dynamic boundary condition** within DualSPHysics. In this way, the fluid sees the flexible structure as a normal moving boundary.

For more information, please see **O'Connor, J. and Rogers, B.D.** ***Journal of Fluids and Structures*, 2021**.

XML file: Special-flexstrucs

Definition of Flexible Structures

```
<commands>
    <mainlist>
        <setdrawmode mode="full" />
        <!--Clamp-->
        <setmkbound mk="0" />
        <drawsphere radius="0.05">
            <point x="0.2" y="0.0" z="0.2" />
        </drawsphere>
        <!--Flexible Structure-->
        <setmkbound mk="1" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0.2" y="-0.01" z="0.19" />
            <size x="0.4" y="0.02" z="0.02" />
        </drawbox>
    </mainlist>
</commands>
```

Geometry definition

```
</geometry>
<motion>
    <objreal ref="1">
        <begin mov="1" start="0" />
        <mvnull id="1" />
    </objreal>
</motion>
```

Tag as moveable object

```
</casedef>
<execution>
    <special>
        <flexstrucs>
            <flexstrucbody mkbound="1" mkclamp="0">
                <density value="1000.0" comment="Mass density" units_comment="kg/m^3" />
                <youngmod value="1.4e6" comment="Young's Modulus" units_comment="Pa" />
                <poissratio value="0.4" comment="Poisson ratio" />
                <constitmodel value="1" comment="Constitutive model 1:Plane Strain (2D),
                <hgfactor value="0.0" comment="Hourglass correction factor: keep as low
            </flexstrucbody>
        </flexstrucs>
    </special>
```

Flexible structure definition

There are three main steps to setting up a case involving flexible fluid-structure interaction:

1) Define the geometry of the flexible structure and where it will be clamped.
2) Tag the flexible structure as a moveable object.
3) Tag the flexible structure and its clamp and define the material and numerical properties.

XML file: Special-flexstrucs

Example 1: Turek & Hron (CSM3)

Geometry Definition

```xml
<mainlist>
    <setdrawmode mode="full" />
    <!--Clamp-->
    <setmkbound mk="0" />
    <drawsphere radius="0.05">
        <point x="0.2" y="0.0" z="0.2" />
    </drawsphere>
    <!--Flexible Structure-->
    <setmkbound mk="1" />
    <drawbox>
        <boxfill>solid</boxfill>
        <point x="0.2" y="-0.01" z="0.19" />
        <size x="0.4" y="0.02" z="0.02" />
    </drawbox>
</mainlist>
```

It is necessary to define the geometry of the clamp first, then embed the flexible structure within the clamp.

Tag as Moveable Object

```xml
<motion>
    <objreal ref="1">
        <begin mov="1" start="0" />
        <mvnull id="1" />
    </objreal>
</motion>
```

The flexible structure is tagged as a moveable object via the **<motion/>** block by setting the **<mvnull/>** label. This tells DualSPHysics that the motion is to be computed during runtime.

XML file: Special-flexstrucs

Example 1: Turek & Hron (CSM3)

Flexible Structure Definition

```
<special>
    <flexstrucs>
        <flexstrucbody mkbound="1" mkclamp="0">
            <density value="1000.0" comment="Mass density" units_comment="kg/m^3" />
            <youngmod value="1.4e6" comment="Young's Modulus" units_comment="Pa" />
            <poissratio value="0.4" comment="Poisson ratio" />
            <constitmodel value="1" comment="Constitutive model 1:Plane Strain (2D),
            <hgfactor value="0.0" comment="Hourglass correction factor: keep as low
        </flexstrucbody>
    </flexstrucs>
</special>
```

mkbound and **mkclamp** are the mkbound numbers for the flexible structure and clamp, respectively.

density is the mass density (kg/m^3) of the flexible structure, **youngmod** is the Young's modulus (Pa), and **poissratio** is the Poisson ratio.

constitmodel is the constitutive model to be used. This defines the stress-strain relationship for the flexible structure. There are two available in 2-D (plane strain and plane stress) and one for 3-D (St. Venant-Kirchhoff). See the end of this document for more details.

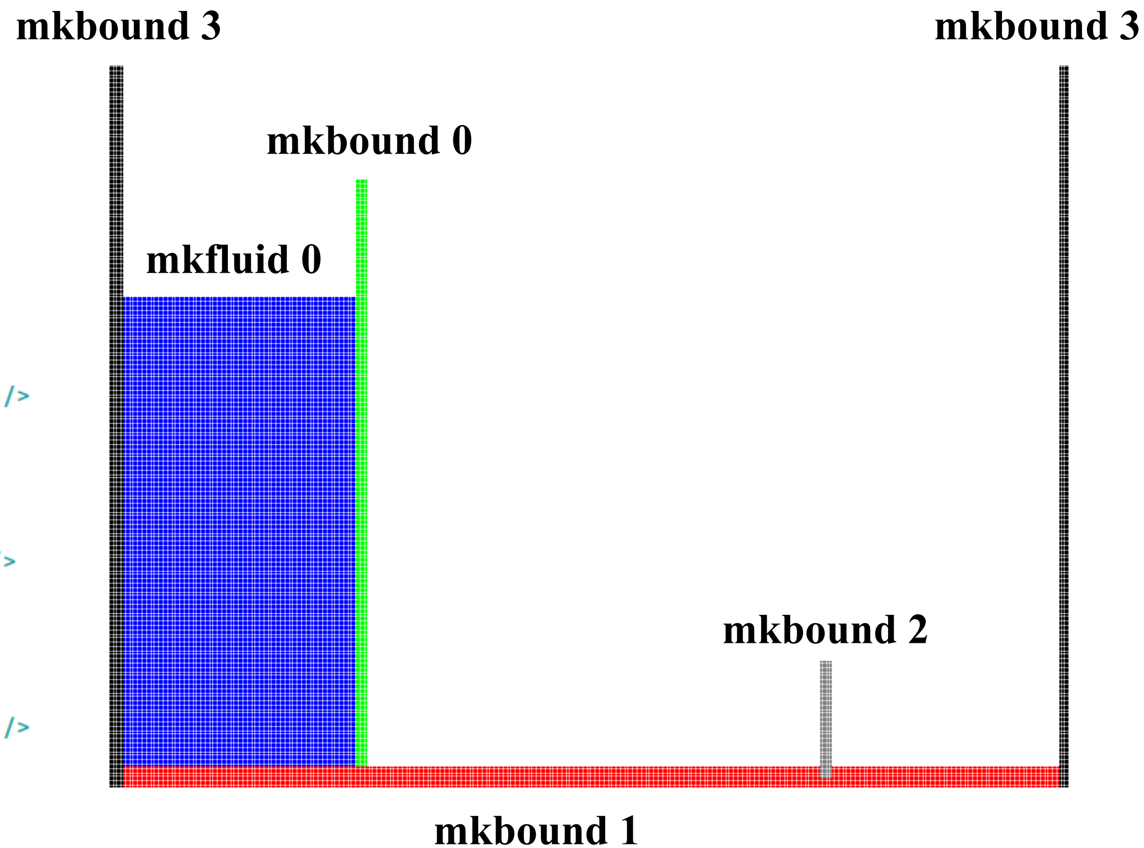
hgfactor is the hourglass correction factor to use in the zero-energy mode suppression scheme. Note that non-zero values will increase the effective stiffness of the flexible structure. Therefore, it is recommended to set this to zero first and only increase it if zero-energy instabilities appear. If required, a value of 0.1 has been found to work effectively with negligible impact on the effective stiffness.

XML file: Special-flexstrucs

Example 2: Dam Break (2D)

Geometry Definition

```
<commands>
    <mainlist>
        <setdrawmode mode="full" />
        <setmkfluid mk="0" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0.0" y="-0.01" z="0.0" />
            <size x="0.2" y="0.02" z="0.4" />
        </drawbox>
        <setmkbound mk="0" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0.2" y="-0.01" z="0.0" />
            <size x="0.008" y="0.02" z="0.5" />
        </drawbox>
        <setmkbound mk="1" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="-0.008" y="-0.01" z="-0.016" />
            <size x="0.816" y="0.02" z="0.016" />
        </drawbox>
        <setmkbound mk="2" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0.596" y="-0.01" z="-0.008" />
            <size x="0.008" y="0.02" z="0.098" />
        </drawbox>
        <setmkbound mk="3" />
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="-0.008" y="-0.01" z="-0.016" />
            <size x="0.008" y="0.02" z="0.616" />
        </drawbox>
        <drawbox>
            <boxfill>solid</boxfill>
            <point x="0.8" y="-0.01" z="-0.016" />
            <size x="0.008" y="0.02" z="0.616" />
        </drawbox>
    </mainlist>
</commands>
```



XML file: Special-flexstrucs

Example 2: Dam Break (2D)

Tag as Moveable Object

```
<motion>
    <objreal ref="0">
        <begin mov="1" start="0.0" finish="0.12" />
        <mvfile id="1" duration="0.12">
            <file name="DambreakGateTimeSeries.csv" fields="2" fieldtime="0" fieldz="1" />
        </mvfile>
    </objreal>
    <objreal ref="2">
        <begin mov="1" start="0" />
        <mvnull id="1" />
    </objreal>
</motion>
```

As well as tagging the flexible structure as a moveable object, a realistic gate motion is also included to initiate the dam break.

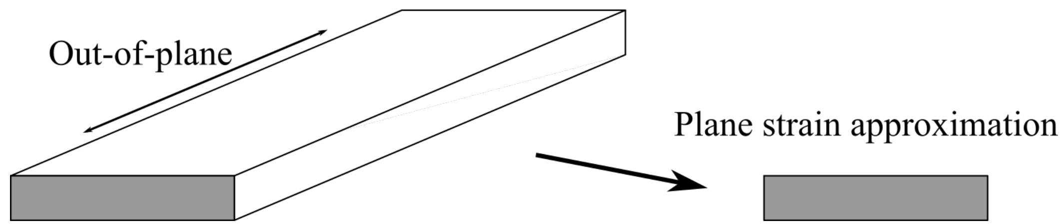
Flexible Structure Definition

```
<special>
    <flexstrucs>
        <flexstrucbody mkbound="2" mkclamp="1">
            <density value="1161.54" comment="Mass density" units_comment="kg/m^3" />
            <youngmod value="8.75e5" comment="Young's Modulus" units_comment="Pa" />
            <poissratio value="0.49" comment="Poisson ratio" />
            <constitmodel value="1" comment="Constitutive model 1:Plane Strain (2D),
            <hgfactor value="0.1" comment="Hourglass correction factor: keep as low a
        </flexstrucbody>
    </flexstrucs>
</special>
```

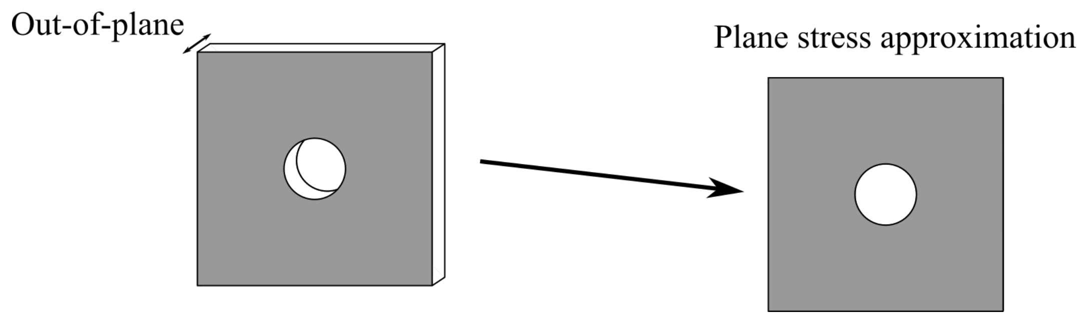
Constitutive Model

There are three possible options for **constitmodel**: 1 (plane strain), 2 (plane stress), and 3 (St. Venant-Kirchhoff).

Plane strain is only valid for 2-D problems and models the third (out-of-plane) dimension by assuming there is **zero strain in the third dimension**. This is suitable for problems that are **very thick in the out-of-plane direction**.



Plane stress is only valid for 2-D problems and models the third (out-of-plane) dimension by assuming there is **zero stress in the third dimension**. This is suitable for problems that are **very thin in the out-of-plane direction**.



The hyperelastic **St. Venant-Kirchhoff** constitutive model is the only available 3-D model and is an extension of the **linear elastic** model to the **geometrically nonlinear regime**.