The MultiCoFEA software

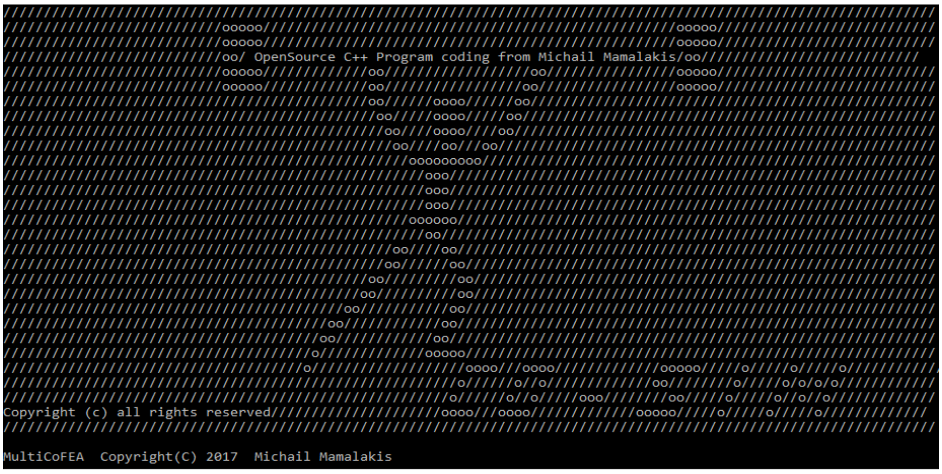

Figure 2.1 The start of the MultiCoFEA software

Overview

For this thesis, it created a cooperation software which has as scope to automate the process of applying the forces or the prescribe motion of each joint of interest from a multibody analysis simulation to a finite element analysis geometry and solve an implicit dynamic analysis of a specific motion of joint.

MultiCoFEA is a framework software which allows the user to generate a finite element analysis .xml file with all the boundary and contact constrained of the two rigid bodies of the geometry joint of interest. This XML file constricts base a multibody analysis (OpenSim software) of a motion of interest.

With a setup .txt file, the user could control the DOFs of motion of the two rigid bones. He could set a forward, an inverse or hybrid dynamic analysis. There are two optional analysis. In first could applied prescribed motions or forces in both rigid bodies, and in the second could applied prescribed motions or forces only in one rigid body and in the second rigid body set it as a ground body and fixed in all dimensions.

The main future of MultiCoFEA is the two kind of analysis (JR_starter and BF_starter), a time resample (ReSampler) of the analysis from multibody data output, a degree of freedom resample mainly for the body forces-motions analysis (DOFResample), a time step founder for the finite element analysis (TSC) and a pre visualizer of the geometry's motion (Visualizerfebgeo).

MultiCoFEA is written in C++. The source code is freely available and open source (https://github.com/ece7048/MultiCoFEA). The software is designed for use in Microsoft Windows operating systems. Among these tools is an IniReader library to read and write txt files (set up .txt configuration file). A user guide is available online and the code is heavily commented to allow users with the C++ knowledge to explore the algorithms and to make changes to the code if desired.

The user could use a specified OpenSim multibody model and run an inverse kinematic-dynamics simulation for a specific joint. After that can set the rotation-translation axis where the two rigid bodies could move for the finite element analysis. Then he could do a time resample of the forces-motions data for reducing the length of the output data. He could selects the kind of analysis where the finite element analysis runs a Joint motion-force analyses or a separate rigid body motion-force applied analysis. Then he could choose in how many time partition want to run the whole simulation of interest and simulated them parallel. After that created the XML .feb file of the finite element method (FEBio software). If the user wants could see a preview motion of the joint which will be simulated from the finite element analysis (FEA). After that, the MultiCoFEA program starts the simulation.

The user must define the OPENSIM_HOME variable and to add OPENSIM_HOME/bin to the path and the FEBio variable and to add FEBio/bin to path too.

MultiCoFEA Workflow

In this section, we will describe the recommended workflow to create a FEBio XML file based the motion and forces results data of multibody software (OpenSim) (Figure 2.5).

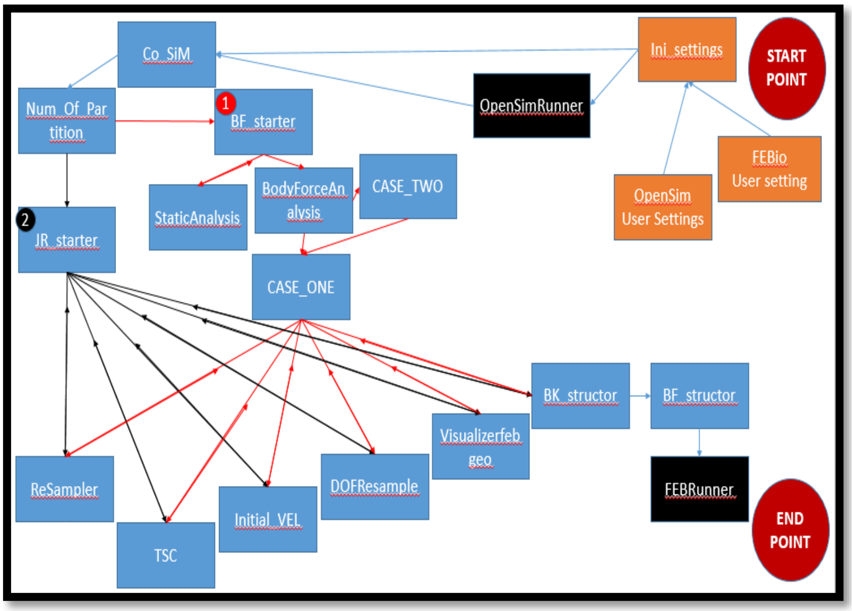

Figure 2.2 The template of the cooperation code between the OpenSim and FEBio.

<u>The Setup file</u>

In the start point, the user has to set up some basic variables which configure the analysis. In Figure 2.6 illustrate these variables. The "CMCanalysis" is a variable for the analysis of OpenSim. If these analyses were already done you set 'y'. In the other case, he have to set the analysis which already mention in OpeSim section and are illustrated in Figure 2.6 (selection 'n'). The "MODEanalysis" is the type of analysis that desire to do the user. If he set "R" the analysis will be a Joint Reaction analysis in OpenSim and the output data will be the six vectors of motions and the six, of reaction forces data of the unlocked joint's degree of freedom of OpenSim's model. If the user set "F" the analysis will be body motions-forces six vectors data that will be translated to the point of interest based the FEA geometry.
After that, the "PartitionIntervals" is the number of the time partitions which the user wish to create. If set "1" then the time's data will keep whole without separation in sub simulations. The "MANUALintervals" is if the user wants to set by own the time intervals of the partitions simulations manually ("y") or let the code do it automatically ("n"). The "MOD_of_interval" is in case the "MANUALintervals" is set in "n" mode and define the kind of degree of freedom (DOF) the user wants to take account the partition inverter. The “r” define the rotation's DOF and "t" the translation's.

Moreover, the "Initialtime" variable is the time interval of the initial analysis of the finite element method to reach the initial pose-forces of the joint geometry. The "Point_of_acting_forces" if it set "d" is the default point which is in the 0, 0, 0 center point for each rigid body (bone) of the joint. If it set "nd then it takes account the "xvalue", "yvalue", "zvalue" which are the coordination of point interest.

```
[BASICSETUP]

CMCanalysis=n

MODEanalysis=R

PartitionIntervals=1
MANUALinterval=n
MOD_of_interval=r

Initialtime=0.2

Point_of_acting_forces=d
xvalue=0
yvalue=0
zvalue=0

Translation_from_JR=y

Interpolation_Detect=n
Interpolation_kind=l

Time_Resample=y
DOF_Resample=y

TSC_method=n
Max_timesteps=0
Min_timesteps=0

Visualizer_premotion=y

Joint_free=n

Forced_DOF_1=NAN

Prescribed_DOF_1=zyx_ryrxrz

Fixed_DOF_1=NAN


Forced_DOF_2=NAN

Prescribed_DOF_2=NAN

Fixed_DOF_2=zyx_ryrxrz

Translation_costrain=n

[PATH]

MODELS = /models/OpenSim/Gait2354_Simbody/sixdof/RRA/results/rramodel2.osim
MODELFeb = /models/FEBio/tf_joint.feb
RESULT_DIR = /data/results
RESULT_DIR2 = /data/results2
PROGRAMMNAME=C:\OpenSim 3.3\bin\
NAME=opensim64.exe
```

Figure 2.3 The basic variables and the paths of MultiCoFEA configuration file.

For the body forces-motions analysis only, the user can set if he wants the translation DOF from the joint analysis for a simpler version problem something like a hybrid analysis between joint and bodies forces-motions detection (set "y").

Next, the user has to set if he wants interpolation detection of the data's behavior by set "y" in "Interpolation_Detect". If this variable set "n", he could choose between two option only set all the interpolation point linear or smooth by set the variable "Interpolation_kind", "l" or "s".

After this follows the "Time_Resample" and "DOF_Resample". The first variable is for doing a resample in time space for reducing the test points. The second one is for doing a DOF resample and set only the DOF which change between the two bodies. The "DOF_Resample" is a mention for only body forces-motions analysis. In case, he wants to set "y" the "Translation_from_JR" variable then the "DOF_Resample" must be "y" too for correct results of forces vectors.

The "TSC_method" refers the time step counter detector which computes the time step that the FEA analysis will have to do. This is an empirical detector and if the user wants could try it by setting the variable "y" and modified the results if he wants after. If the "TSC_method" set "y" then he has to set the max and min time step that the user wants.

The "Visualizer_premotion" refers the preview motion which implements in FEA. The "Joint_free" is a variable for cases where the user wants to set a rotation DOF prescribed and set another rotation DOF unconstrained. If he wants something like that he has to set "y" this variable.

After that follows the DOF strategy for the FEA of the two rigid bodies. The user has to set the DOF that wants to applied forces ("Forced_DOF") and the DOF that wants to apply prescribed motion ("Prescribed_DOF") and the fixed DOFs. If a DOF is not set in these three variables it will be unconstrained.

Finally, the "translation_costrain" is a mode variable where when is set "y" the user can set a translation threshold and change these data. In "PATH" section the user has to set the paths for OpenSim model after the RRA analysis ("MODELS"), the Febio's geometry model

("MODELFeb"), the Result direction ("RESULT_DIR"), the program name of OpenSim and the executable file of them. In Figure 2 it illustrates an example of these variables setting.

```
[STATIC]
STATE =C:/Users/ece/Desktop/Co_SiM_withFEBiocoding/models/OpenSim/Gait2354_Simbody/sixdof/ST/results/stmotion.mot
START_TIME =0
END_TIME =0.1

[BODYFORCES]
STATE = /models/OpenSim/Gait2354_Simbody/sixdof/CMC/results/subject01_walk1_states.sto
BDNAME1= femur_r
BDNAME2= tibia_r
JOINT=knee_r
JOINT1=knee_tz_r
JOINT2=knee_tx_r
JOINT3=knee_ty_r
JOINT4=knee_angle_r
JOINT5=knee_add_r
JOINT6=knee_rot_r
POINT0=0
START_TIME =0.427
END_TIME =1.615

[FEBIOSTEP]
GEO=/models/FEBio/GEO
GEOF=.\femur.obj
GEOS=.\tibia.obj

PROGRAMMNAME=C:\Program Files\febio-2.5.0\bin\
NAME=FEBio2.exe

TYPE1=solid2
TIMESTEPS1=20
STEPSIZE1=1e-006
DTOL1=0.01
ETOL1=0.02
RTOL1=0.1
LSTOL1=0
PRES_STIF1=0
DTMIN1=0
DTMAX1=0
MAX_RET1=10
OPT_ITER1=100
AGGRES1=1
MAX_REF1=100
MAX_UPS1=0
OPT_BW1=0
RESTART1=0
PLOT_LEVEL1=PLOT_MUST_POINTS
CMAX1=1e+5
ANALYSIS1=static
alpha1=0
beta1=0
gamma1=0
PRINT_LEVEL1=PRINT_MINOR_ITRS
MINRESIDUAL1=1e-15
OUT1=OUTPUT_MAJOR_ITRS
```

Figure 2.4. The static, body forces and Febio analysis variables and the paths of MultiCoFEA configuration file.

In Figure 2.7 someone could obtain the "STATIC", "BODYFORCES" and "FEBIOSTEP" sections. In the "STATE" variable the user has to set the path for the states of the static analysis of the OpenSim simulation. The "BDNAME1" is the name of the first rigid body and the "BDNAME2" is for the second. In "JOINT" the user set the virtual joint's name of interest, which is define in OpenSim without the coordination sign (tx, ty, tz, rot etc.). In "JOINT1" until "JOINT6" the user set the unlock joints of OpenSim's model with first the translations and then the rotation DOF. The "START_TIME" and "END_TIME" is the start and end time of analysis respectively.

In "FEBIOSTEP" section the user set variables depends the FEBio analysis. "GEO" is the path for the GEO folder where the user could direct the two rigid bodies object files for the preview illustration ("GEOF", "GEOS").The other Variables is depend the FEBio analysis set up which already mention in previously in Control section.

In Figure 2.8 observed the OpenSim's analysis which have to already done from the user. If is not the user has to set the paths for the set up files ("SETUP"), the OpenSim's model ("MODEL"), a result direction ("RESULT_DIR"), a start and end time, plus some specific files which OpenSim's individual analysis need.

```
[INVERSEKINEMATICSTATIC]
MODEL=/models/OpenSim/Gait2354_Simbody/sixdof/ST/subject01_simbody.osim
MARKER = /models/OpenSim/Gait2354_Simbody/sixdof/ST/subject01_static.trc
SETUP= /models/OpenSim/Gait2354_Simbody/sixdof/ST/subject01_Setup_ST.xml
RESULT_DIR = /models/OpenSim/Gait2354_Simbody/sixdof/ST/results
START_TIME = 0
END_TIME = 0.1

[INVERSEKINEMATICS]
MODEL=/models/OpenSim/Gait2354_Simbody/sixdof/IK/subject01_simbody.osim
SETUP= /models/OpenSim/Gait2354_Simbody/sixdof/IK/subject01_Setup_IK.xml
RESULT_DIR = /models/OpenSim/Gait2354_Simbody/sixdof/IK/results
GRFNAME=subject01_walk1
START_TIME = 0
END_TIME = 2.1

[RRA]
RESULT_DIR = /models/OpenSim/Gait2354_Simbody/sixdof/RRA/results
SETUP= /models/OpenSim/Gait2354_Simbody/sixdof/RRA/subject01_Setup_RRA.xml
TASK=/models/OpenSim/Gait2354_Simbody/sixdof/RRA/gait2354_RRA_Tasks.xml
ACTUATOR=/models/OpenSim/Gait2354_Simbody/sixdof/RRA/gait2354_RRA_Actuators.xml
CONTROL=/models/OpenSim/Gait2354_Simbody/sixdof/RRA/gait2354_RRA_ControlConstraints.xml
EXFORCES=/models/OpenSim/Gait2354_Simbody/sixdof/RRA/subject01_walk1_grf.xml
START_TIME =0.397
END_TIME =1.626

[CMC]
RESULT_DIR = /models/OpenSim/Gait2354_Simbody/sixdof/CMC/results
SETUP= /models/OpenSim/Gait2354_Simbody/sixdof/CMC/subject01_Setup_CMC.xml
MOTION= /models/OpenSim/Gait2354_Simbody/sixdof/RRA/results/subject01_walk1_RRA_Kinematics_q.sto
TASK=/models/OpenSim/Gait2354_Simbody/sixdof/CMC/gait2354_CMC_Tasks.xml
ACTUATOR=/models/OpenSim/Gait2354_Simbody/sixdof/CMC/gait2354_CMC_Actuators.xml
START_TIME =0.397
END_TIME =1.626
```

Figure 2.5 The OpenSim analysis variables and the paths of MultiCoFEA configuration file.

After the set up file the user has nothing to do he has to run only the executable file.

<u>The code workflow</u>

In Figure 2.5 someone could obtain the headers of the MultiCoFEA software. The main source code is the Co_SIM. This is the function for reading the setup file and start the processes. Based the simulation that already set the user will follow the path 2 (for joint reaction analysis) or path 1(for body analysis). The paths are illustrated in Figure 2.8 with red and black vectors respectively. To begin with it will examined separate every path (red, black) started from the second.

If the user selects the Joint Reaction analysis then the code follows the second path (black vectors) in Figure 2.5.

The "JR_starter" is the handler of the analysis which based the configuration setup file will call or not the individual sources. These sources are common for both simulations. The "JR_starter" runs the JointReaction analysis of OpenSim for the DOF data. The "ReSampler" is the source function where run a resample of time vector, which based on the slope's change of every degree of freedom that participates in the analysis. After the detection of time values where the slope change for every DOF, it updates the time vector with all the time values that occurred in all degrees of freedom for having a homogenous time vector. The "TSC" is a time step detector for the FEBio analysis. it takes account the desired maximum and a minimum number of steps that the user wants and with a linear interpolation detect the number of main analysis based the steps of the initial analysis (the analysis of the initial pose and forces of the virtual joint). The "Initial_VEL" is an initial velocity writer based theOpenSim analysis which returns the initial velocity of the degree of freedom that the user set for a specific analysis. If the "DOFResaple" is used then it calls the initial velocity source code to resample the DOF velocities too. The "Visualizerfebgeo" is responsible for writing the OpenSim files for the preview motion visualization. It gives the user the opportunity to change or to write new motion's data for each DOF he wants. For any change that he did in DOFs space of the joint, the software calls the "DOFResample" to correct the forces vectors. The "BK_structor" and the "BF_structor" write the final form of the degrees of freedom that will use the finite element analysis, the FEBio software. These source codes take account the section of DOF strategy that already mentioned in setup file before. Finally called the "FEBRunner" for connecting all the written in text form files and creates the final FEBio XML file. Depends on the partition number that the user set in a setup file the process loop all over again and creates as much FEBio's XML files as the partition number is.

The "DOFResample" is a source code which manipulates the degrees of freedom mainly in order of the body analysis (BodyForceAnalysis) but and of the joint analysis (JR_starter) in some cases. The analysis of this function present below.

If the user select the Body analysis then follows the red vector of Figure 2.5. The "BF_starter" is the handler of the analysis.

The "BF_starter" calls the "BodyForceAnalysis" and do an analysis where detect the motion and the acting forces in the two rigid bodies of interest in the point which the user had set in the setup file. The Static analysis ("StaticAnalysis") is used to remove the DC component vector of global coordination frame from each of the position vectors. Finally, the "CASE_TWO" and "CASE_ONE" are called and handle all the already mention source codes that are in common with the Joint Reaction analysis.

DOF_Resample mathematical analysis

Generally, the initial velocity, the motion and the external forces defined as input in the finite element models. From the OpenSim could set each human’s virtual joint (between the two bodies of interest in our case femur and tibia) from one to six degree of freedom. Let assume that is set in one degree of freedom (DOF). Base on that it will estimate the muscle forces which act in two bodies based the one degree of freedom (an unreal case, in real life a body has six DOF). The bodies can move in the six degrees of freedom in OpenSim analysis because is unconstrained in the six dimensional space. The “unlock” virtual joint of OpenSim leaves only one DOF to move so the two bodies at the end will have different motion only in this particular DOF (in the other will have the same amount of motion). The “DOF Resample” is a class that resamples the DOF space of virtual joint, based the different values of the six spatial vectors between the two body positions and corrects the forces and initial velocity vectors. Base the [49] and the Plücker notation matrix is already know that the way to transform acting velocities or forces from one point to another is the equation (1) and (2) which obtain below.

Let **A** and **B** be Cartesian frames with origins at O and P, respectively, let **r** be the coordinate vector expressing *OP* in **A** coordinates, and let **E** be the rotation matrix that transforms 3D vectors from **A** to **B** coordinates. The Plücker transform from **A** to **B** coordinates is then the product of a translation by **r** and a rotation by **E.** The formula for a motion-vector transform is:

$$^{B}X_{A} = \begin{vmatrix} E & 0 \\ 0 & E \end{vmatrix} \begin{vmatrix} 1 & 0 \\ -r\times & 1 \end{vmatrix} = \begin{vmatrix} E & 0 \\ -Er\times & E \end{vmatrix} \tag{1}$$

The equivalent transform for a force vector is:

$$^{B}X_{A}^{*} = \begin{vmatrix} E & 0 \\ 0 & E \end{vmatrix} \begin{vmatrix} 1 & -r\times \\ 0 & 1 \end{vmatrix} = \begin{vmatrix} E & -Er\times \\ 0 & E \end{vmatrix} \tag{2}$$

Where r x = is a 3x3 matrix with values:

$$\begin{pmatrix} x \\ y \\ z \end{pmatrix} \times = \begin{vmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{vmatrix} \tag{3}$$

Based these equations could be transformed a six dimension vector from a point to another. The "DOFResampler" uses these transforms matrixes to correct the forces vector and the initial velocity vector. The matrix transforms the vectors of interest from the initial acting point (in OpenSim simulation) to a new acting point with ***fix degrees of freedom*** (these which are same between the two rigid bodies) and ***unfixed the DOF*** (these which are different). In this way, could be reduced the DOFs of the model in only them which actually change.

The mathematically way for this implementation is to calculate a matrix based the (1), (2) in which the rotation degrees will take the negative values of the ***fix degrees of freedom*** of two bodies and the ***unfixed DOF*** will take zero value in all time steps of OpenSim's simulation. With this procedure, the force and initial velocity vector corrects in two new vectors, based the new motion of bodies. For example, if it is detected only one DOF which change between the two bodies and let assume that is a rotation in the x-axis, then it will compute a transform matrix based the angles of y rotation axis, z rotation axis and the distances of x, y, z translation axis. Finally, the rotation x-axis sets zero in all time steps. The rotation matrix **E** of equation (1) and (2) takes the form of the Tait-Bryan [51], for the z, x, y rotation-axis, based the coordination global axis of FEBio's software. This matrix is observed below:

$$\mathbf{Z_1X_2Y_3} = \begin{vmatrix} c1c2 - s1s2s3 & -c2s1 & c1s3 + c3s1s2 \\ c3s1 + c1s2s3 & c1c2 & s1s3 - c1c3s2 \\ -c2s3 & s2 & c2c3 \end{vmatrix} \tag{4}$$