Core Transport Simulator (ETS)

December 17, 2020

Contents

1 ETS source in FORTRAN

You can checkout the FORTRAN ETS workflow from gforge [1] / project ETS [2] following instructions from ETS User Guide [3]

If you did not use ETS before, first you need to request access to the code via the EFDA ITM Portal [4]by following the GForge tab, following the project ETS [5]and requesting access.

Once you have access to the code, it can be checked out of SVN using

```
svn co https://gforge6.eufus.eu/svn/ets
```

to access the whole repository, or

```
svn co https://gforge6.eufus.eu/svn/ets/trunk/ETS
```

to access just the trunk version of the ETS.

The ETS project on Gforge [6] also includes:

- A wiki [7]
- Some documentation [8]
- Trackers [9]
- News [10]
- Mailing lists [11]
- The SVN repository (web interface) [12]

[1] https://www.efda-itm.eu/ITM/html/itm_glossary.html#g_gforge
[2] https://gforge6.eufus.eu/project/ets/
[3] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_User_Guide.pdf
[4] https://gforge6.eufus.eu/
[5] https://gforge6.eufus.eu/project/ets/
[6] https://gforge6.eufus.eu/project/ets/
[7] https://gforge6.eufus.eu/project/ets/wiki/
[8] https://gforge6.eufus.eu/project/ets/docman/
[9] https://gforge6.eufus.eu/project/ets/tracker/
[10] https://gforge6.eufus.eu/project/ets/news/
[11] https://gforge6.eufus.eu/project/ets/mailman/
[12] https://gforge6.eufus.eu/project/ets/scmsvn/

2 ETS workflows in KEPLER

The ETS workflow is used for 1-D transport simulation of a tokamak core plasma.

ETS workflows in KEPLER :

- use actors and composite actors from other IMPs, thus for the most recent versions of them please check with relevant project
- complex, but clearly structured workflow, which offers user friendly interface for configuring the simulation
- allow for easy modifications (*connecting new modules, or reconnecting parts of the workflow*) through an easy graphical interface
- provide users with all updates through the version control system
- still in active development tool

There are currently 2 workflows being developed within ITM-IMP3 project:

- ETS_A_4.10b [13] Contact person: denis.kalupin@efda.org?subject=ETS%20in%20KEPLER (Skype: dkalupin) (Status) [14]
- ETS_A_4.10a [15] Contact person: denis.kalupin@efda.org?subject=ETS%20in%20KEPLER (Skype: dkalupin) (Status) [16]
- ETS_C [17] Contact person: vincent.basiuk@cea.fr?subject=ETS%20in%20KEPLERphilippe.huynh@cea.fr?subject=ETS% [18]

2.1 ETS_A 4.10b

2.1.1 Obtaining the ETS

Contact person: denis.kalupin@euro-fusion.org?subject=ETS%20in%20KEPLER (Skype: dkalupin)

2.1.1.1 Installing the ETS

The default ETS release is the tag 4.10b10.3 [19]

**Before installation make sure that:*

- you have your private data base for the version of the UAL [20] required by the workflow
- you have the version of KEPLER [21] required by the workflow installed. Quick start on kepler required for the ETS can be found here [22]
- inside the window, where you will be downloading the ETS the source command:

```
        >source $ITMSCRIPTDIR/ITMv1
\textit{Kepler\_Version Data\_Base\_Name UAL\_Version
```

[13] https://www.efda-itm.eu/ITM/html/ETS_A_KEPLER_4.10b.html
[14] https://www.efda-itm.eu/ITM/html/ets_status.html
[15] https://www.efda-itm.eu/ITM/html/ETS_A_KEPLER_4.10a.html
[16] https://www.efda-itm.eu/ITM/html/ets_status.html
[17] https://www.efda-itm.eu/ITM/html/ETS_C_KEPLER.html
[18] https://www.efda-itm.eu/ITM/html/ets_status.html
[19] https://www.eufus.eu/documentation/ITM/html/ETS_A_4.10a_obtain.html#ETS_A_4.10a_obtain_3
[20] http://portal.efda-itm.eu/twiki/bin/view/Main/UalListOrReleases
[21] http://portal.efda-itm.eu/twiki/bin/view/Main/KeplerListOfReleases
[22] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallMultipleKeplers

is executed.

To install your local copy of the ETS workflow please do:

```
>svn co https://gforge6.eufus.eu/svn/keplerworkflows/tags/ets_4.10b10.3/ETS
>cd ETS
>make import_ets
```

Press the play button on the workflow.

The workflow shall run! If it does not, please use the denis.kalupin@euro-fusion.org?subject=ETS%20in%20KEPLER from above.

Starting the workflow: If you have the workflow already installed, there are there are several ways tio execute it:

- For execution via kepler GUI:

  ```
  >kepler.sh workflow_path/workflow_name.xml
  ```

- For execution in none GUI mode:

  ```
  >kepler.sh -runwf -nogui -redirectgui $ITMHOME/some_dir_name workflow_path/workflow_name.xml
  ```

- For execution in batch mode:

  it is essential to keep the workflow inside your $ITMWORK area

  it is essential to switch to scripts/R2.2 module

  ```
  >module switch scripts/R2.2
  >submit_batch_kepler.sh run_dircetory 1 $ITMWORK/workflow_path/workflow_name.xml $ITMSCRIPTDIR/ba
  ```

2.1.1.2 ETS revisions

Revision Name:	*UAL version:*	*KEPLER version:*	*Short Sumary:*	*Comments:*
4.10b0.1	4.10b8_R2.1.0	any, up to 4.10b3.5	Contains:Fixed boundary equlibrium; Simple transport models; full HCD package; Impurity; Pellets; Sawtooth	Test 4.10b release, restricted module choice, restricted physics capabilities, work around of coredelta
4.10b8.1	4.10b8_R2.1.0	central installation 4.10b3_central [23] is preferred; local installation 4.10b3.6 [24] or above	Contains:Fixed boundary equlibrium; Simple transport models; full HCD package; Impurity; Pellets; Sawtooth; Scenario	Test 4.10b release, restricted module choice, restricted physics capabilities, work around of coredelta, produces scenario output on request
4.10b10.1	4.10b10	central installation 4.10b3_central [25] is preferred; local installation 4.10b3.6 [26] or above	MODIFICATIONS COMPATIBLE WITH 4.10b10 DATA STRUCTURE	UNDER CONSTRUCTION: release at the Code Camp in Prague

[23] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler
[24] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler
[25] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler
[26] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler

Revision Name:	*UAL version:*	*KEPLER version:*	*Short Sumary:*	*Comments:*
4.10b10.2	4.10b10_branches_R2.1.r1380	central installation 4.10b3_central [27] is preferred; local installation 4.10b3.6 [28] or above	Added synchrotron radiation, some of neoclassical actors,reworked combiners	UNDER CONSTRUCTION: release at the Code Camp in Prague
4.10b10.3	4.10b10_branches_R2.1.r1380	central installation kepler_rc (2.4/R3.8/kepler or more recent) [29] is preferred	Added synchrotron radiation, some of neoclassical actors,reworked combiners	compared to previous shall contain compeeted transport, new controller for pellet and sawteeth module

2.1.2 Configuring the ETS run

2.1.2.1 WORKFLOW PARAMETERS

2.1.2.1.1 General Parameters

- USER - your userid
- MACHINE - machine name (database name) for which comutations are done
- SHOT_IN - input shot number
- RUN_IN - input run number
- SHOT_OUT - output shot number
- RUN_OUT - output run number
- NUMERICAL_SOLVER - choice of the numerics solving transport equations (RECOMENDED SELECTION: 3 or 4)

2.1.2.1.2 Space resolution

- NRHO - number of radial points for transport equations
- NPSI - number of points for equilibrium 1-D arrays
- NEQ_DIM1 - number of points for equilibrium 2-D arrays, first index
- NEQ_DIM2 - number of points for equilibrium 2-D arrays, second index
- NEQ_MAX_NPOINTS - maximum number of points for equilibrium boundary

2.1.2.1.3 Time resolution

Start and End time:

- TBEGIN - Computations start time
- TEND - Computattions end time

[27] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler
[28] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler
[29] http://portal.efda-itm.eu/twiki/bin/view/Main/InstallANewVersionOfKepler

Time step:

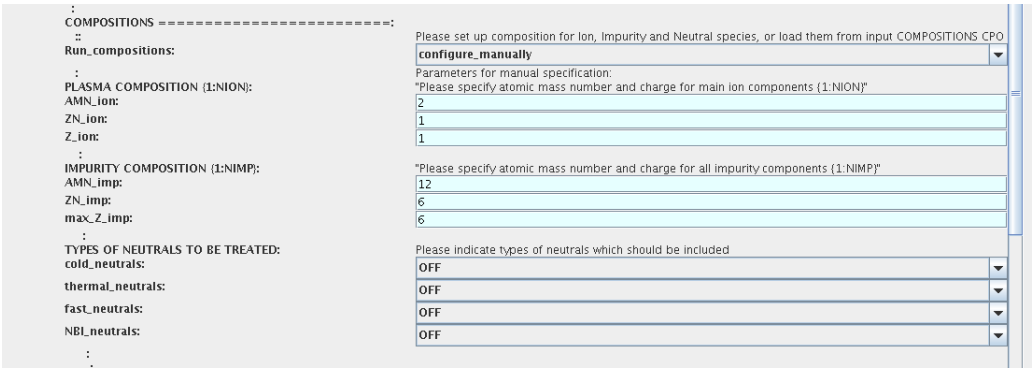
- right click on the box **BEFORE THE TIME EVOLUTION**
- select *Configure actor*
- TAU :specify value of the time step in [s]
- TAU_OUT : specify value of the output time interval in [s]
- *Commit*

2.1.2.2 ION, IMPURITY and NEUTRAL COMPOSITION

Before starting the run you need to define types of main ions, impurity (optional) and neutrals (optional) to be included in simulations.

To define plasma composition:

- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Configure actor***
- choose one of modes for setting *Run_compositions*
  - from_input_CPO - will pick up the COMPOSITIONS structure of the COREPROF CPO saved to the input shot;
  - configure_manually - will force the composition from the values specified below
- specify values of atomic mass (AMN_ion), nuclear charge (ZN_ion) and charge (Z_ion , from the first ion to the last [1:NION] , separated by commas
- (optional) specify values of atomic mass (AMN_imp), nuclear charge (ZN_imp) and maximal ionization state (max_Z_imp) for impurity ions, from the first to the last [1:NIMP] , separated by commas
- (optional)for neutrals activate, by switchen them to ***ON*** , the types which shall be followed by neutral solver
- press ***Commit***



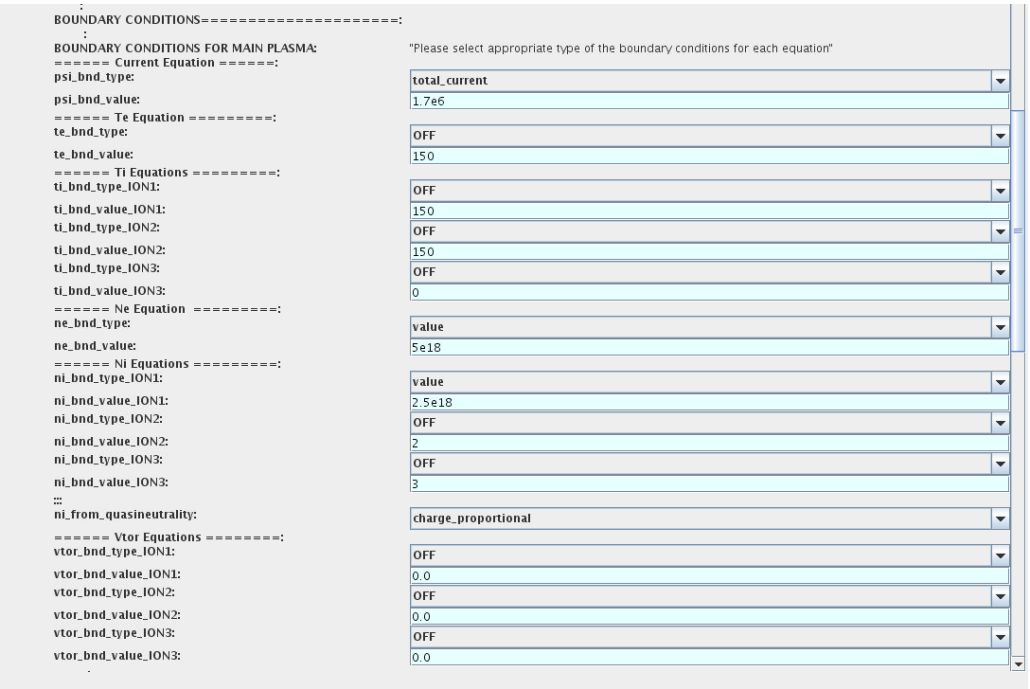
2.1.2.3 EQUATIONS TO BE SOLVED AND BOUNDARY CONDITIONS

2.1.2.3.1 Main Plasma

Before starting the run you need to select the type and value of the boundary conditions for all equations. Please note that the value should correspond to the type. All equations allow for following types of boundary conditions:

- *OFF* - equation is not solved, initial profiles will be kept for whole run
- *value* - edge value should be specified
- *gradient* - edge gradient should be specified
- *scale_length* - edge scale length should be specified
- *generic* - generic form: a1*y + a2*y = a3 of the boundary condition is assumed, 3 coefficients (a1 , a2 , a3) should be provided
- *value_from_input_CPO* - equation is solved, edge value evolution will be red from input shot
- *profile_from_input_CPO* - equation is not solved, profile evolution will be red from input shot

The particular equation will be activated if the boundary condition type for it is other than *OFF*



To set up boundary conditions:

- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Configure actor***
- select appropriate boundary condition for each equation
- specify values for boundary conditions corresponding to the type and to the ion component
- ***Commit***

The workflow will not allow the user all particle components (ions[1:NION]+electrons) to be run predictively. At least one of them shall be set to OFF (this component will be computed from quasi-neutrality condition).

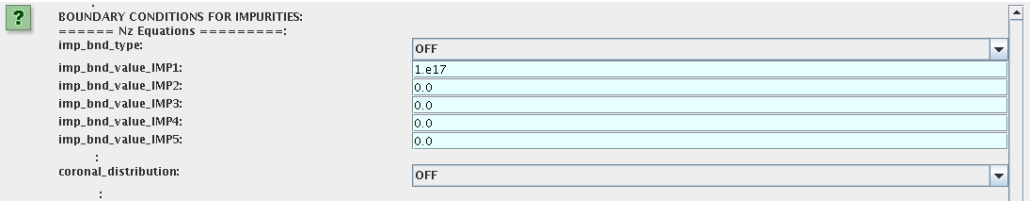
!!! If electron density is solved, all ions with ni_bnd_type=OFF will be computed from the quasineutrality condition and scaled proportional to specified *ni_bnd_value* or inversely proportional to their charge, *charge_proportional* . This is defined by option: *ni_from_quasineutrality* .

2.1.2.3.2 Impurity

You can set up the boundary conditions for impurity ions in a similar way as for main ions.
!!! Note, that at the moment only types: *OFF* ; *value* and *value_from_input_CPO* are accepter by impurity solver.

To set up boundary conditions:

- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Configure actor***
- select appropriate boundary condition for each impurity species (OFF -equation is not solved)
- specify values for boundary density of each impurity component [1:MAX_Z_IMP] , separated by commas
- ***Commit***



Interface for impurity boundary condition has additional option, *coronal_distribution* , that allow to preset the edge values or entire profiles of individual ionization states from coronal distribution. In tis case only single value is required to be specified for each impurity boundary value.
The options are:

- *OFF* - the boundary values for impurity densities will be as they are specified above;
- *boundary_conditions* - the boundary densities will be renormalized with corona, using the first element from above as a total density
- *boundary_conditions_and_profiles* - the boundary densities and starting profiles will be renormalized with corona, using the first element from above as a total density

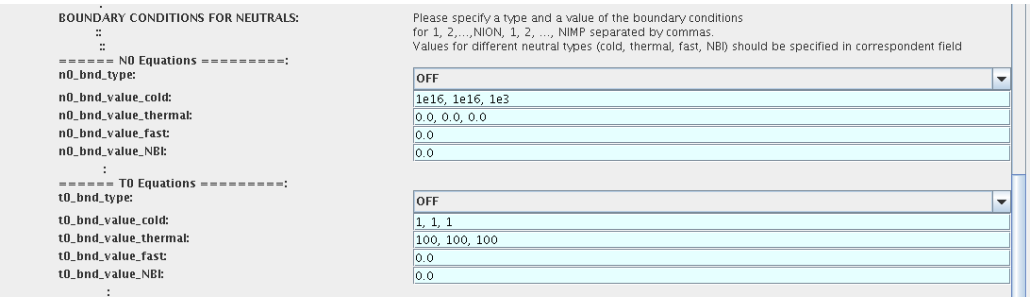
2.1.2.3.3 Neutrals

!!! AT THE MOMENT BOUNDARY CONDITIONS FOR NEUTRAL VELOCITIES ARE DISABLED, MIGHT BE ADDED ON REQUEST

Note, that ALL values should be specified in the order: { *1, 2, 3 ...NION, 1, 2, 3, ...NIMP* }

To set up boundary conditions:

- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Configure actor***
- select appropriate boundary condition for each neutral species (OFF -equation is not solved)
- specify values for boundary density and temperature of each neutral component [1, 2, 3 ...NION, 1, 2, 3, ...NIMP] , separated by commas
- ***Commit***



2.1.2.3.4 Input profiles interpolation

You are going to start the ETS run from some input shot, which might contain some conflicting rho grids saved to different CPOs.
Thus there is a choice for the user to decide on the grid on which the starting profiles should be load by the worflow, *Interpolation_of_input_profiles* .

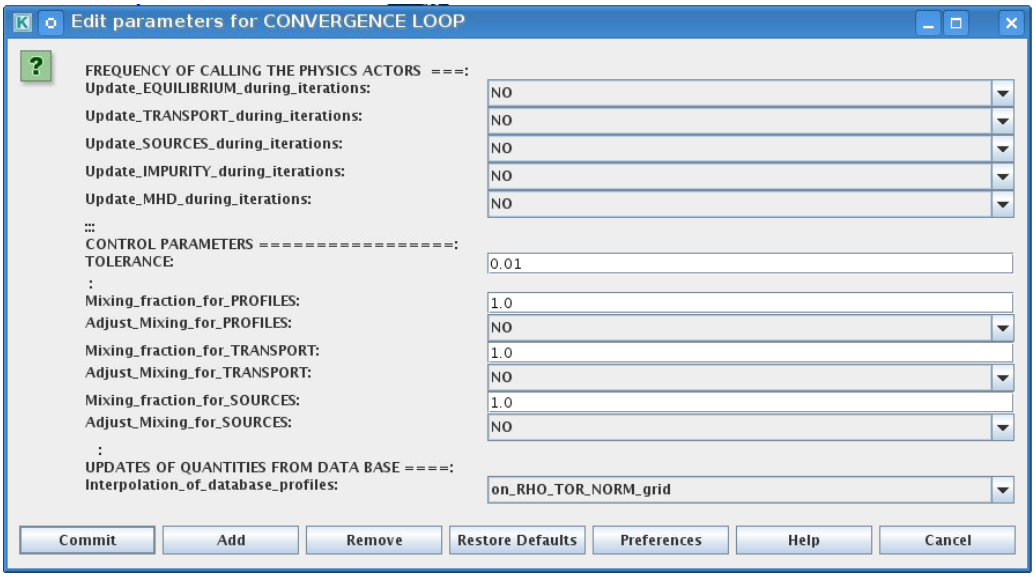
To define the interpolation grid select:

- *on_RHO_TOR_grid* - interpolate input profiles based on the grid specyfied in [m];
- *on_RHO_TOR_NORM_grid* - interpolate input profiles based on normalised rho grid [0:1]

2.1.2.4 CONVERGENCE LOOP

ETS updates input from different physics actors in a sequence, which is finished by solving the transport equations. Ther are possible none-linear couplings between different parts of the system. These nonelinearities are trited by the ETS using iterations.
The decision to step in time is made by the ETS based on the criteria that the maximum relative deviation of main plasma profiles is lower than some predefined tolerance.
There is a number of settings and sitches in the ETS that are used by the iterative scheme. To edit them do:

- right click on the box **CONVERGENCE LOOP**
- select ***Configure actor*** to edit settings
- choose your settings
- ***Commit***



Switches in the field *FREQUENCY OF CALLING THE PHYSICS ACTORS* define how many times the the actors of a certain cathegory (equilibrium, transport, etc.) should be called in a single time step.
By selecting ***YES*** all actors of this cathegory will be called every iteration
By selecting ***NO*** all actors of this cathegory will be called only ones in a time step

Switches and parameters in the field *CONTROL PARAMETERS* define how iterations are done

- *Tolerance* - defines the maximum relative error of profiles change compared to previous iteration. If it is achieved the time steping is done.

For highly none-linear case the required precision can be achieved faster by the iterative scheme if only fraction of the new solution is mixed to the previous state.
The following scheme is adopted by the ets to reduce none-linearities in profiles, transport coefficients and sources:

```
Y = (Amix * Y+) + ((1-Amix)*Y-)
```

where Amix is the mixing fraction

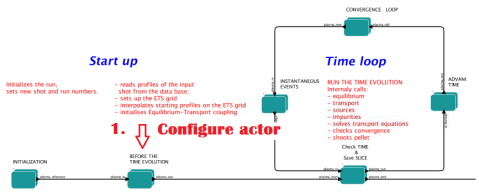
You can activate the mixing of profiles, transport coefficient and sources by selecting the corresponding *Mixing_fraction_...* to be between [0:1]
You also can activate the authomatic ajustment of this fraction by selecting: *Ajust_Mixing_for_...* to ***YES***

2.1.2.5 EQUILIBRIUM

2.1.2.5.1 Initialization Settings

Before starting the run you need to set up your initial equlibrium. There are several options to do it: if your input shot contains the consistent equilibrium with all necessary parameters - you can start immediately from it; if your input shot contains the equilibrium but it is not consistent or some parameters are missing you can check it automatically; if your input equilibrium is corrupt or not present - you can define the starting equlinbrium by tree moment description. To select your starting equilibrium please do:

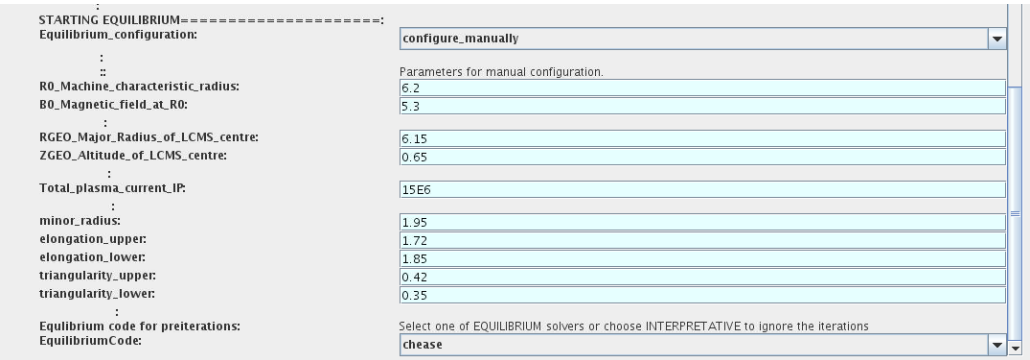
- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Configure actor*** to edit settings
- Select your settings or specify values
- ***Commit***



SETTINGS:

- *Equilibrium_configuration* - select *configure_manually* if you like to specify configuration below; select *from_input_CPO* if all quantities should be picked up from the input CPO
- *R0_Machine_characteristic_radius* - Characteristic radius of the machine, here B0 is measured [m]
- *B0_Magnetic_field_at_R0* - Magnetic field measured at the position R0 [T]
- *RGEO_Major_Radius_of_LCMS_centre* - R coordinate of the geometrical centre of the LCMS [m]
- *ZGEO_Altitude_of_LCMS_centre* - Z coordinate of the geometrical centre of the LCMS [m]
- *Total_plasma_current_IP* - plasma current within the LCMS [A]
- *Minor_radius* - minor radius of the LCMS [m]
- *Elongation* - elongation of the LCMS [-]
- *Triangularity_upper* - upper triangularity of the LCMS [-]
- *Triangularity_lower* - lower triangularity of the LCMS [-]

- *Equilibrium code* - select one of available equilibrium solvers to check the consistency between starting equilibrium and current profile; use *INTERPRETATIVE* if you trust your input data (in this case the check will be ignorred).

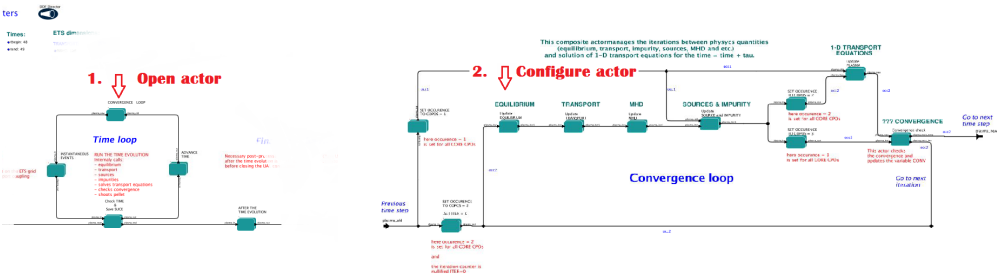


Please note, that different equilibrium solvers might require slightly different input. Thus it is a user responsibility to check that the information inside input shot/run is enough to run selected equilibrium solver.

2.1.2.5.2 Run Settings

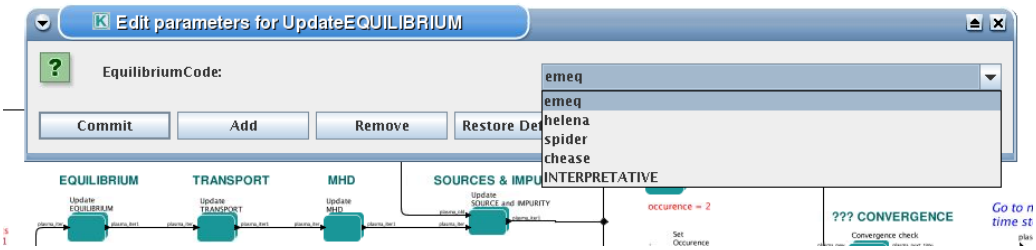
There are several equilibrium solvers connected to the ETS. You can select the one of them.Therefore please do:

- right click on the box **CONVERGENCE LOOP**
- select ***Open actor***
- right click on the box **EQUILIBRIUM**
- select ***Configure actor*** to edit settings
- choose your equilibrium solver
- ***Commit***



INTERPRETATIVE means that the ETS will not update the equilibrium, instead it will be using the initial equilibrium.

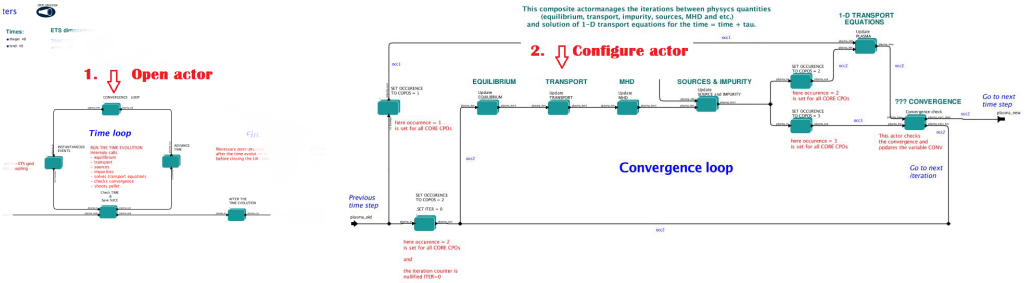
Please note, that it is better to select the same code as you used for pre-iterrations. Because outputs of different equilibrium solver are not necessary done with the same resolution. Therefore the routine saving the information to the data base might brake due to uncompatible sizes of some signals.



2.1.2.6 TRANSPORT

The settings for TRANSPORT can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

- right click on the box **CONVERGENCE LOOP**
- select ***Open actor***
- right click on the box **TRANSPORT**
- select ***Configure actor*** to edit settings
- choose your settings
- press ***Commit***



2.1.2.6.1 Transport models

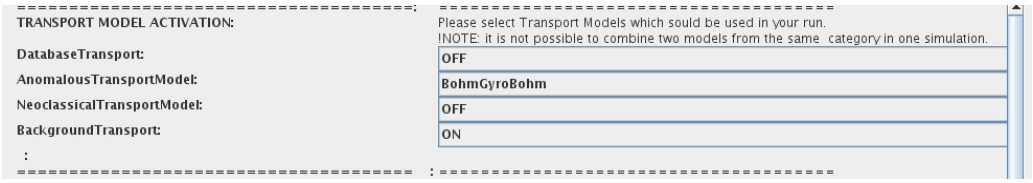
ETS constructs the total transport coefficients from the combination of Anomalous transport (model choice), Neoclassical transport (model choice), Database transport (transport coefficients be saved to the input shot) and Background transport (Transport coefficients defined through the GUI interface)

D_tot = D_DB*M_DB + D_AN*M_AN + D_NC*M_NC + D_BG*M_BG

You should choose from the list of available models in each cathegory or switch it ***OFF***

Individual multipliers for all channels shall be specified on the lower level through the code parameters of **Transport Combiner**

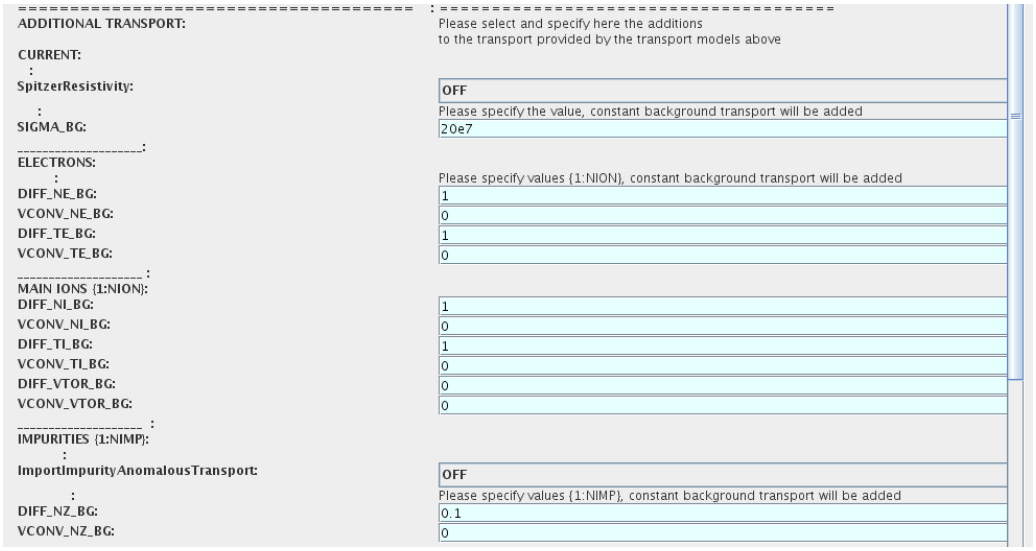
The list of available transport models can be found here [30].



2.1.2.6.2 Background transport

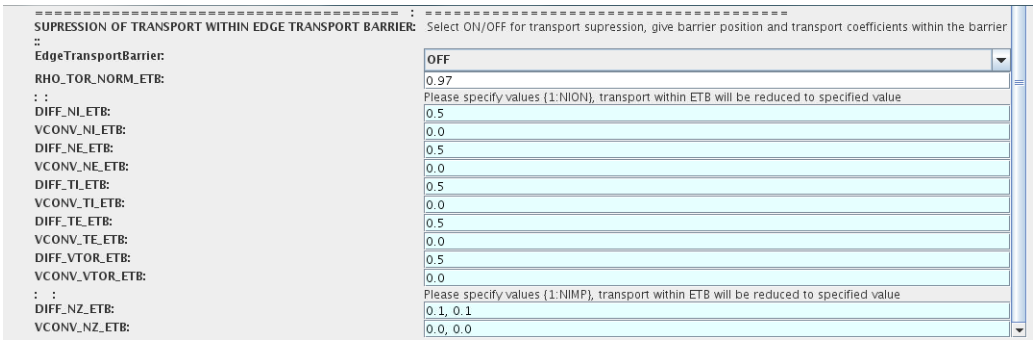
You can add the constant background level for each coefficient (ion and impurity coefficients are expected to be the strings of [1:NION] and [1:NIMP] elements respectively, separated by commas)

[30] https://www.eufus.eu/documentation/ITM/html/ets_status.html



2.1.2.6.3 Edge transport barrier

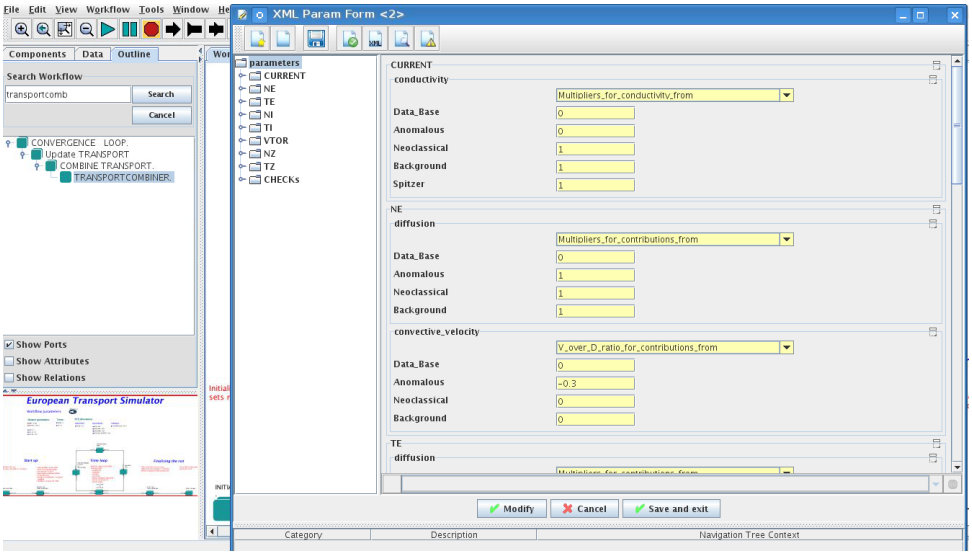
In this section you can artificially supress the transport outside of specified *RHO_TOR_NORM_ETB* . Total transport coefficients for all transport channels (ne, ni, nz, Te, Ti,...) will be reduced to constant values specified below (ion and impurity coefficients are expected to be the strings [1:NION] and [1:NIMP] respectively)



2.1.2.6.4 Total transport coefficients

The fine tuning of of transport coefficients can be done through editing the XML code parameters of the ***transport combiner*** actor:

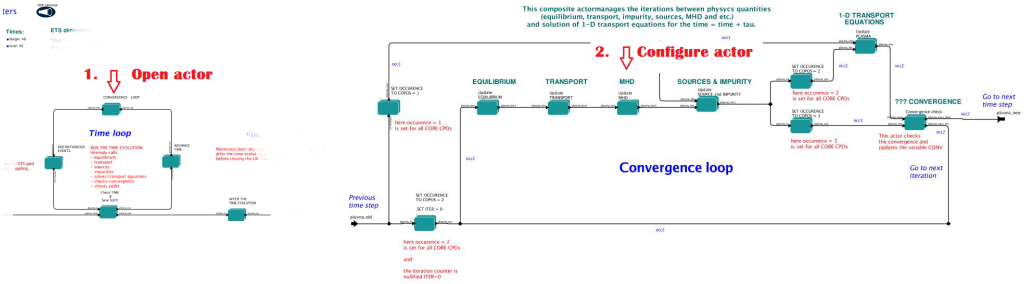
- In *Outline* browse for **transportcombiner**
- select ***Configure actor***
- click ***Edit Code Parameters***
-
  - If you select ***OFF*** contributions from all transport models to this channel will be nullified;
  - If you select ***Multipliers_for_contributions_from*** the transport channel will be activated, and the total transport coefficient will be combined from active tranport models. You gust need to specify multiplier against each channel;
  - For convective velocity there is an additional option ***V_over_D_ratio_for_contributions_from*** . With this option selected the combiner will ignore the convective components provided by transport models. The convective velocity will be determined from the diffusion coefficient by applying fixed V/D ratio (*for inward pinch the values should be negative!*).
- ***Save and exit***
- ***Commit***



2.1.2.7 MHD

The settings for MHD type of events can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

- right click on the box **CONVERGENCE LOOP**
- select ***Open actor***
- right click on the box **MHD**
- select ***Configure actor*** to edit settings
- choose your settings
- ***Commit***

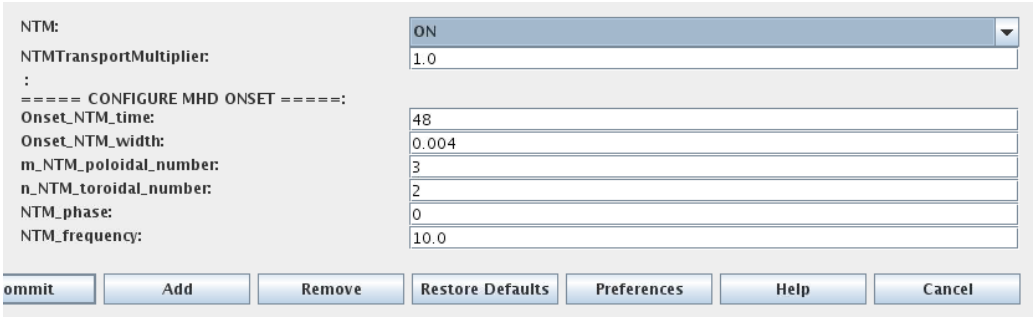


At the moment ETS allows only for NTM to be activated. The sawtooth module is expected to be deployed before ITM Code Camp in Slovenia.

User can ajust the following NTM settings:

- NTM ***ON*** means that ETS will add the NTM driven transport to the total transport coefficient; ***OFF*** -ignored
- NTMTransportMultiplier the transport contrinution from NTM will be multiplied with this value
- Onset_NTM_time - activation time for the NTM mode
- Onset_NTM_width - starting width of the mode
- m_NTM_poloidal_number
- n_NTM_toroidal_number
- NTM_phase

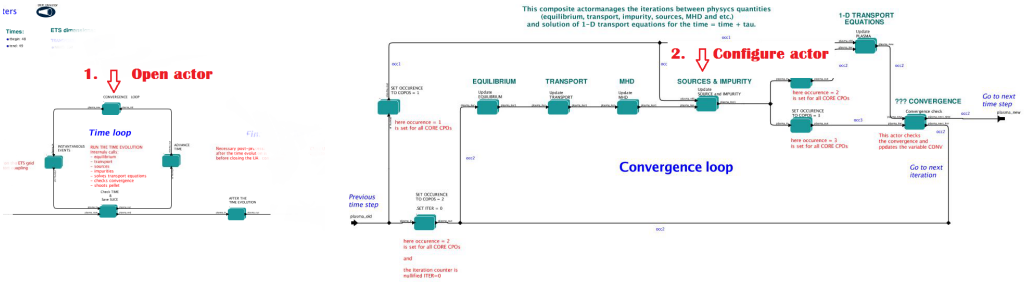
- NTM_frequency



2.1.2.8 SOURCES AND IMPURITY

The settings for SOURCES AND IMPURITY can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

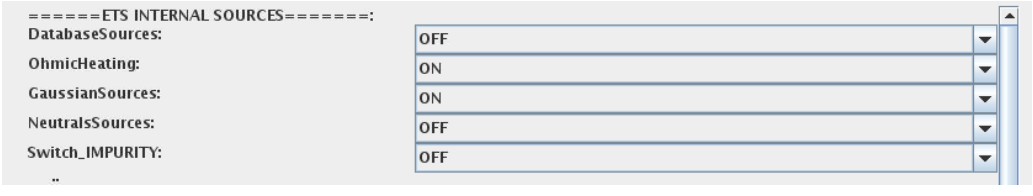
- right click on the box **CONVERGENCE LOOP**
- select ***Open actor***
- right click on the box **SOURCES AND IMPURITY**
- select ***Configure actor*** to edit settings
- choose your settings
- ***Commit***



2.1.2.8.1 Analytical & Impurity sources

There is a number of sources developed by IMP3 project, which are actors or internal routines of the transport solver. You can activate them by selecting ***ON*** / ***OFF*** in front of corresponding source:

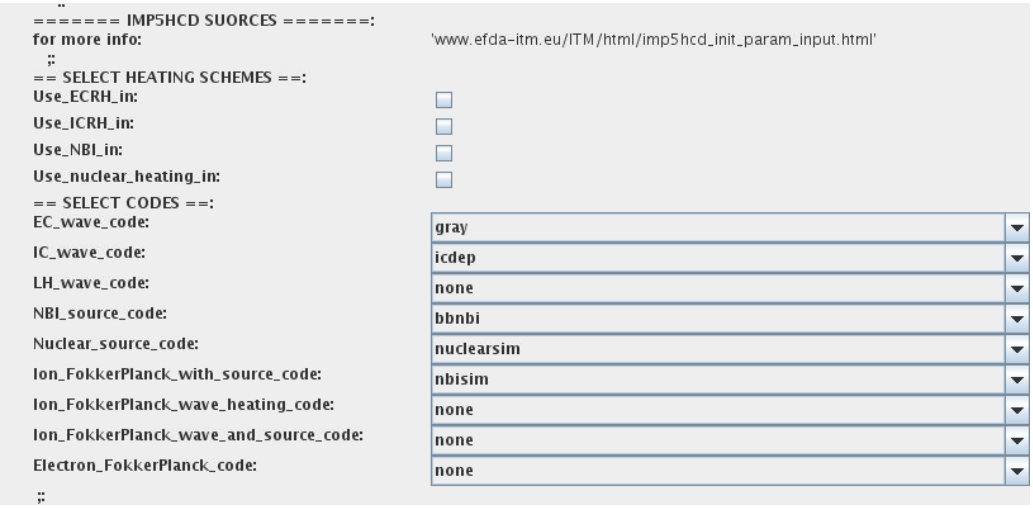
- Database Sources ***ON*** - ETS will pick up the evolution of source profiles saved to your input shot/run; ***OFF*** -ignored
- Ohmic Heating ***ON*** - ETS will compute Ohmic heating internaly; ***OFF*** -ignored
- Gaussian Sources ***ON*** - ETS will add sources from the Gaussian source actor (you can configure heat and particle deposition profiles by editing the code parameters of the actor); ***OFF*** -ignored
- Neutral Sources ***ON*** - Fluid neutrals will be solved according to the boundary conditions specified on Before_time_evolution composite actor interface; ***OFF*** -ignored
- Switch_IMPURITY ***ON*** - Impurity density and radiative sources will be computed; ***OFF*** -ignored; ***INTERPRETATIVE*** profiles of impurity density will be read from input shot/run



2.1.2.8.2 HCD sources

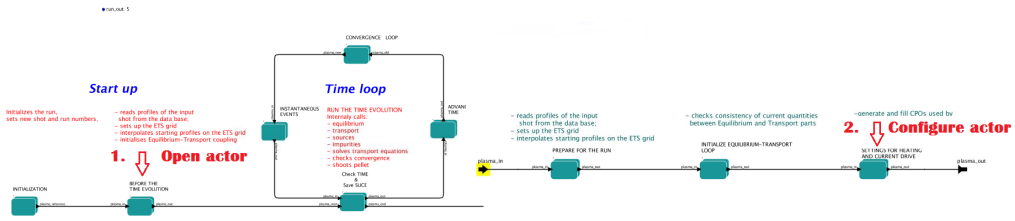
There is a number of sources developed by HCD project, that are incorporated by the ETS workflow.

For the HCD sources please activate the type of heating source, by ticking the box in front of it, and select the code to simulate it.



You also need to configure initial IMP5HCD settings. Therefore please:

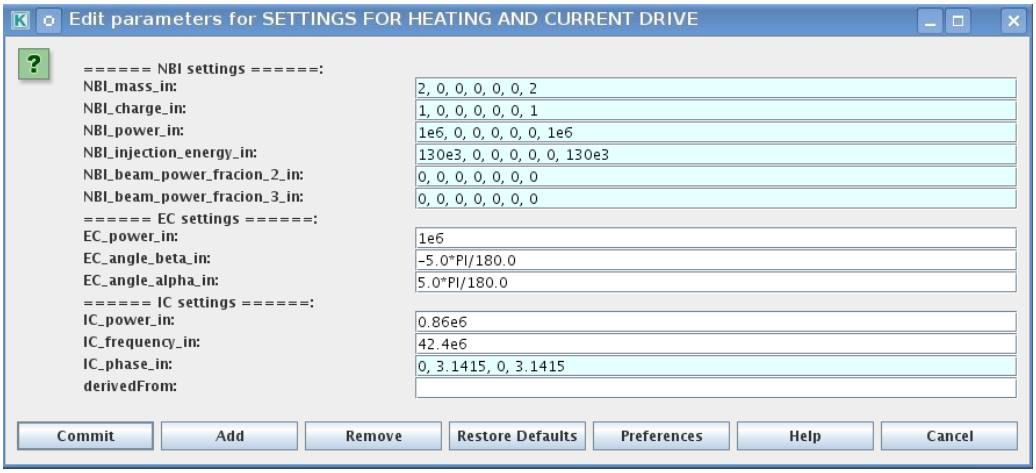
- right click on the box **BEFORE THE TIME EVOLUTION**
- select ***Open Actor***
- right click on the box **SETTINGS FOR HEATING AND CURRENT DRIVE**
- select ***Configure actor***
- edit the stettings
- ***Commit***



The detailed information on initial IMP5HCD settings can be found here [31].

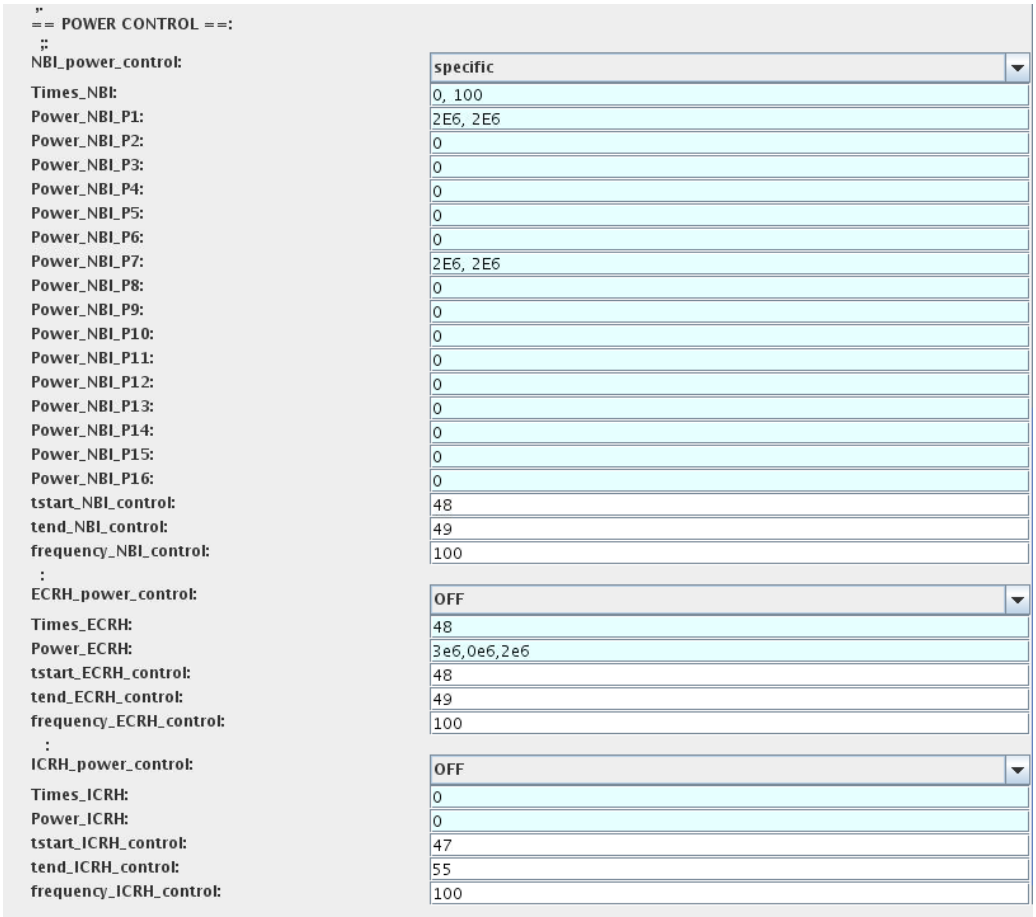
Please note that settings for NBI are done independent for each PINI. Therefore, for NBI settings, please insert the values separated by commas. The number of the element in the array corresponds to the number of activated PINI. Maximum accepted number of PINIs = 16.

[31] https://www.eufus.eu/documentation/ITM/html/imp5_imp5hcd.html



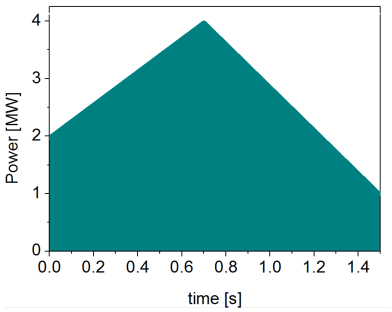
2.1.2.8.3 Power control

You also can activate the power control for the IMP5HCD sources.

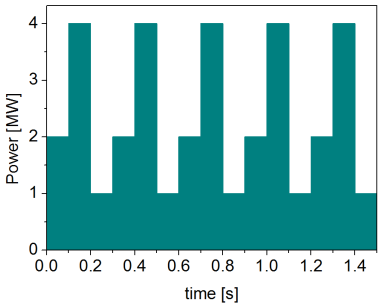


If the POWER_CONTROL is not ***OFF*** , there are two modes of operation: ***specific*** and ***frequency***

For ***specific*** you should specify the time sequence separated by commas and the corresponding power sequence (where first power level corresponds to the first time, second to second and etc.). Linear interpolation will be done between the sequence points.
For example: if you give the power ***sequence*** = 2e6,4e6,1e6 and ***times*** = 0.0, 0.7, 1.5 (s) the delivered power would be:



For ***frequency*** you should specify the power levels sequence separated by commas, start and end time of the power control and the frequency of switching between these levels.
For example: if you give the power ***sequence*** = 2e6,4e6,1e6 and ***frequency*** = 10 (Hz) ***tstart*** = 0.0 (s) ***tend*** = 1.5 (s) the delivered power would be:



2.1.2.8.4 Total power

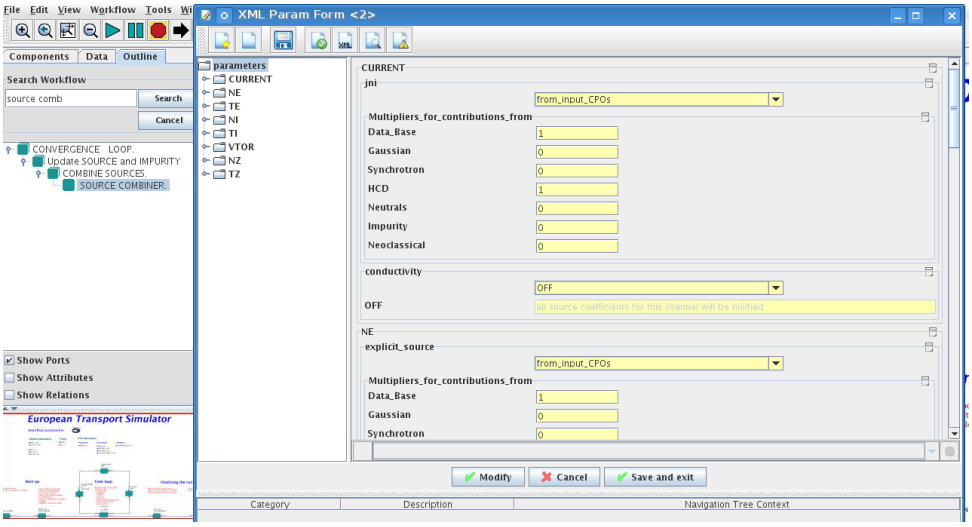
Profiles of the total source for each channel are obtained from the the individual contributions (Data Base, Gaussian, Neutrals, Impurity and HCD) as a summ of all activated sources multiplied with coefficients specified on the interface of the composite actor.

S_tot = S_DS*DSM + S_GS*GSM + S_Neu*NeuSM + S_IMP*IMPSM + S_HCD*HCDSM

The fine tuning of of sources can be done through editing the XML code parameters of the source combiner actor:

- In the *Outline* browse for **source combiner**
- select ***Configure actor***
- click ***Edit Code Parameters***
- If you like the sources to the particular equation being activated - select ***from_input_CPOs*** , and then, put the multipliers against each contribution; if you select ***OFF*** contributions from all sources to this channel will be nullified.

- save and exit
- ***Commit***



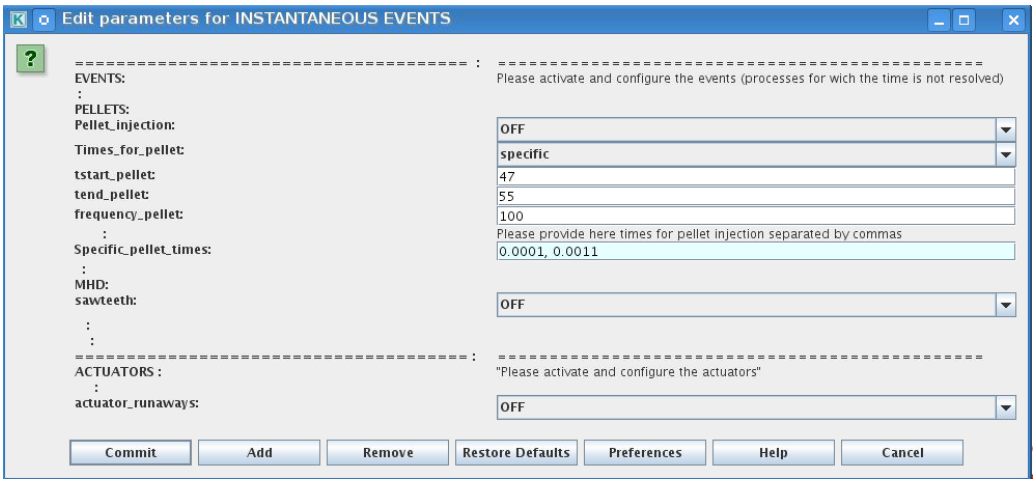
2.1.2.9 INSTANTANEOUS EVENTS & ACTUATORS

At the moment, user can swith ***ON*** and ***OFF*** two types of events: PELLET and SAWTOOTH

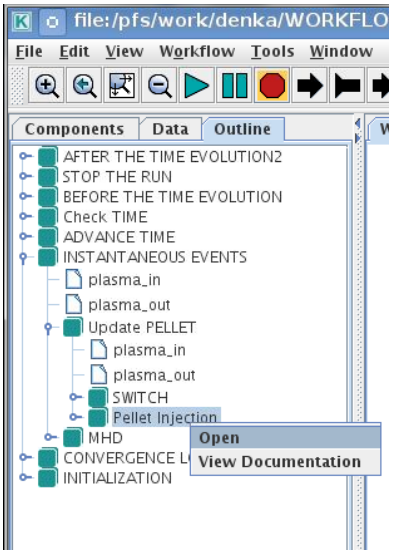
2.1.2.9.1 Pellet

At the top level of the workflow you can configure times for pellet injection

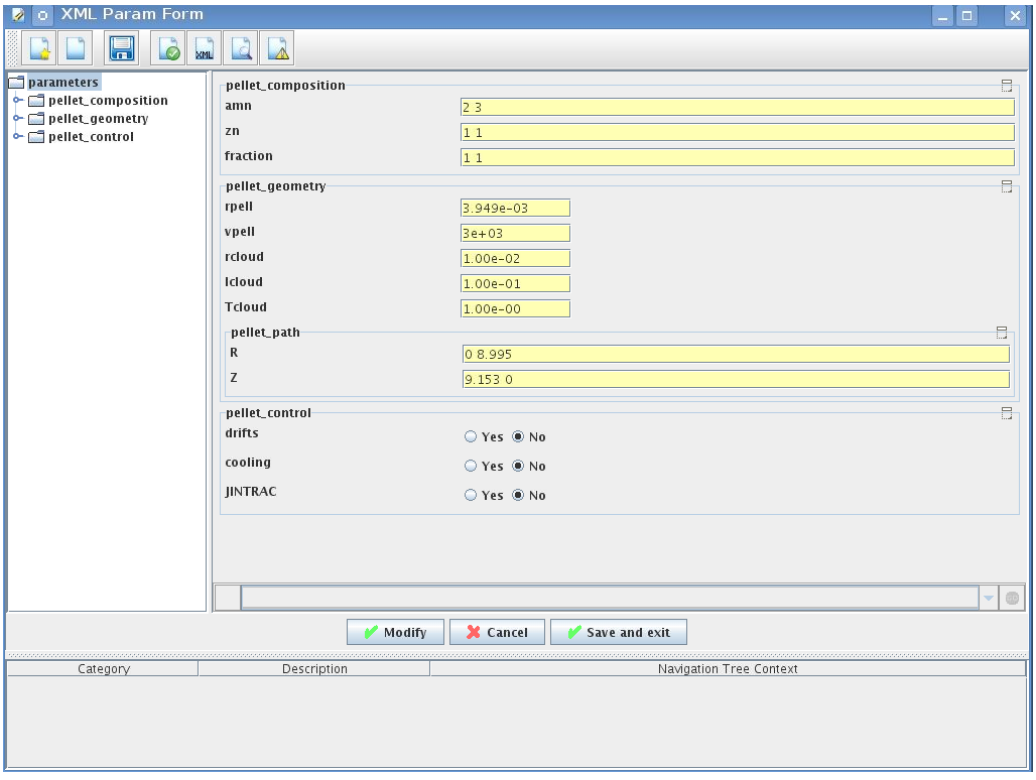
- right click on the box **INSTANTANEOUS EVENTS & ACTUATORS**
- select ***Configure actor*** to edit settings
- Select Pellet_injection equal ***ON*** if you like to use pellet in your simulation
- Select mode of operation:
  - Times_for_pellets equals ***specific*** - pellets will be shut at exact times specified in array times_pellet
  - Times_for_pellets equals ***frequency*** pellets will be shut from tstart_pellet until tend_pellet with a frequency_pellet
- ***Commit***



Parameters of individual pellet need to be configured through the code_parameters of the PELLET actor. To access it go to ***Outline*** on the right upper corner and open the following:



- right click on the actor **PELLET**
- select ***Configure actor***
- click ***Edit Code Parameters***
- edit parameters and click ***save and exit***
- ***Commit***



amn atomic mass number: array of elements separated by space (1:nelements) [-]

zn nuclear charge: array of elements separated by space (1:nelements) [-]

fraction fraction of each element in the pellet, based on the number of atoms: array of elements separated by space (1:nelements) [-]

rpell radius of the pellet [m]

vpell velocity of the pellet [m/s]

rcloud radius of the pellet cloud [m], radial extension of the cloud = 2*rp0

lcloud length of the pellet cloud along the field line [m]

Tcloud temperature of the pellet cloud [eV]

Pellet path is specified by two points, for which R and Z coordinated should be specified

R R coordinates of the pivot and second points of the pellet path, separated by space [m]

Z Z coordinates of the pivot and second points of the pellet path, separated by space [m]

Control switches allow to activate:

- drifts - YES - will activate radial displacement of deposition profile, same for all path points
- cooling - YES - will activate cooling of the other side of the plasma due to parallel heat transport (essential for large pellets, which might cross the same flux surface twice)
- JINTRAC - YES - will provide temperature reduction consistent with the model used in JETTO

2.1.2.9.2 Sawtooth

At the top level of the workflow you can switch ON/OFF possible MHD events

- right click on the box **INSTANTANEOUS EVENTS & ACTUATORS**
- select ***Configure actor*** to edit settings
- Select SAWTOOTH ***ON*** if you like to use them in your simulation
- ***Commit***

2.1.2.9.3 Actuators

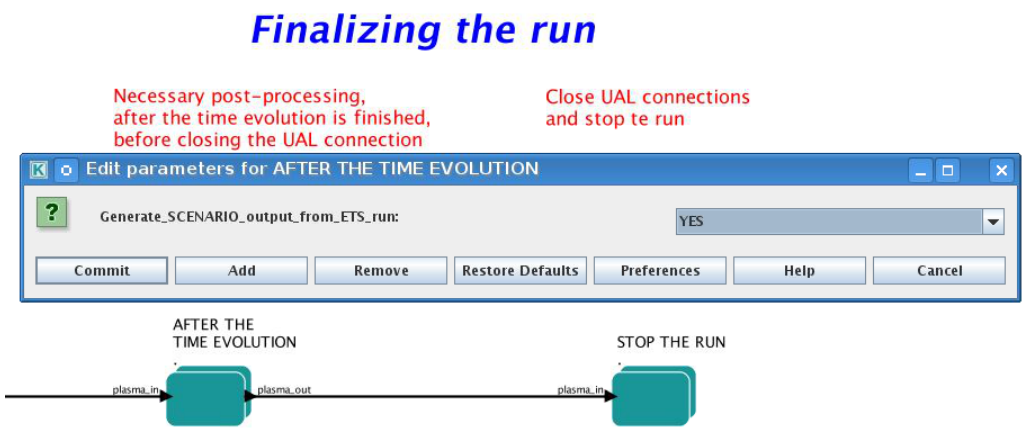
At the top level of the workflow you can switch ON/OFF actuator for runaways

- right click on the box **INSTANTANEOUS EVENTS & ACTUATORS**
- select ***Configure actor*** to edit settings
- Select actuator_runaways ***ON*** if you like to use them in your simulation
- ***Commit***

2.1.2.10 SCENARIO OUTPUT

You can summarize the ETS run by activating the output to SCENARIO CPO (as post-processing of the run).
To activate the SCENARIO output:

- right click on the box **AFTER THE TIME EVOLUTION**
- select ***Configure actor***
- select Generate_SCENARIO_output_from_ETS_run equal ***YES***
- ***Commit***



2.1.2.11 VISUALIZATION

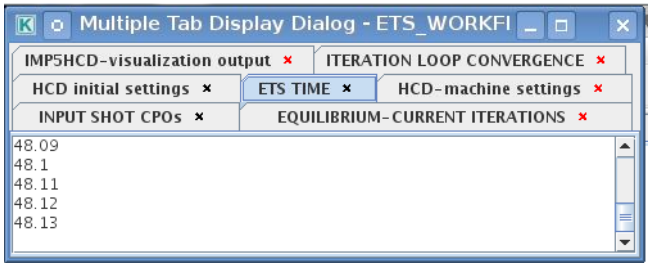
There is a number tools visualizing the ETS run.

2.1.2.11.1 Multiple Tab Display

The display appeares automaticaly when the ETS workflow is launched. It displays diagnostic text messages from the workflow on following topics:

- Input data statement
- Iterations to check the initial convergence between EQUILIBRIUM and CURRENT
- Time evolution
- Convergence of iteratinos within the time step
- IMP5HCD settings
- Power used by IMP5HCD actors durung the run

Also the error messages from execution of the workflow will be displayed here.



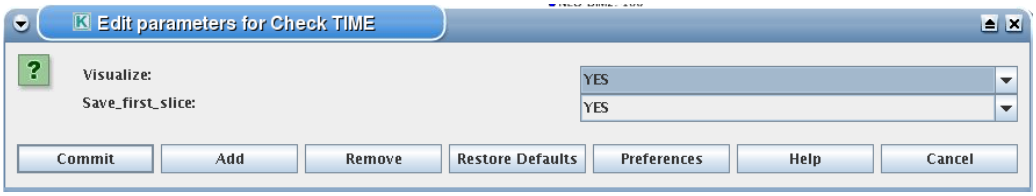
2.1.2.11.2 Python Visualization Display

Please note, if you plan to use python based vizualization ***nomatlab*** argument is essential by the opening of the workflow.

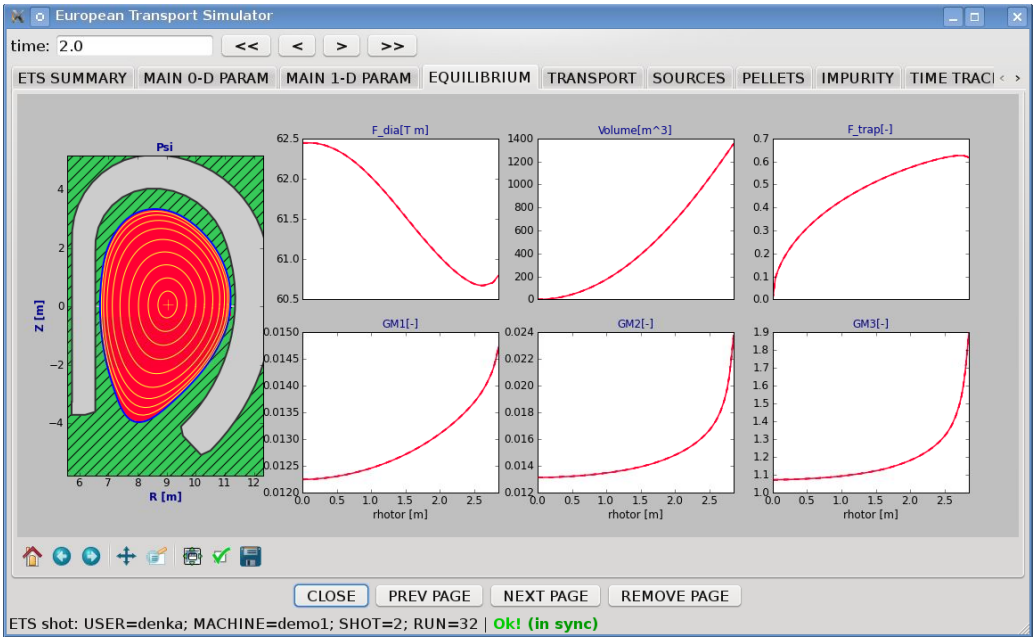
```
>kepler.sh nomatlab workflow_path/workflow_name.xml
```

You can activate the graphical visualization of your run evolution:

- right click on the box **Check Time & Save Slice**
- select ***Configure actor***
- select visualisation ***YES*** or ***NO***
- ***Commit***



Then evolution of main discharge parameters will be shown in this window:



2.1.3 LIST OF ACTORS

UNDER DEVELOPMENT

2.1.3.1 Equilibrium actors

Code name	Code Category	Contact persons	Short description
chease	Grad-Shafranov solver	Olivier Sauter	Chease is a fixed boundary Grad-Shafranov solver based on cubic hermitian finite elements see [H. Lütjens, A. Bondeson, O. Sauter, Computer Physics Communications 97 (1996) 219-260 [32]]
emeq	-	-	
spider	-	-	

2.1.3.2 Core transport actors

Code name	Code Category	Contact persons	Short description
ETS	Transport solver	Denis Kalupin	
BohmGB	Bohm/gyro-Bohm transport coefficients	-	
TCI/Weiland	Transport coefficient from drift wave turbulence	Pär Strand	
TCI/GLF23	Transport coefficient from drift wave turbulence	-	
TCI/RITM	Transport coefficient from drift wave turbulence	-	
TCI/MMM (not yet in ETS)	Transport coefficient from drift wave turbulence	-	
TCI/EDWM (not yet in ETS)	Transport coefficient from drift wave turbulence	-	

[32] https://crppwww.epfl.ch/~sauter/chease/Lutjens_CHEASE_CPC96.pdf

Code name	Code Category	Contact persons	Short description
nclass (not yet in ETS)	Neoclassical transport coefficients	Pär Strand	
neos (not yet in ETS)	Neoclassical transport coefficients	Olivier Sauter	
neowesz	Neoclassical transport coefficients	Bruce Scott	Neoclassical transport coefficients based on the expression in John Wesson's book Tokamaks.
neoartz	Neoclassical transport coefficients	Bruce Scott	
spitzer			
ETBtransport			
coronal			
synchrotronsources			

2.1.3.3 Edge transport actors

2.1.3.4 Heating and current drive actors

Code name	Code Category	Contact persons	Short description
gray [33]	EC/waves	Lorenzo Figini	GRAY is a quasi-optical ray-tracing code for electron cyclotron heating & current drive calculations in tokamaks. Code-parameter documentation can be found here [34].
travis [35]	EC/waves	Nikolai Marushchenko and Lorenzo Figini	Travis is a ray-tracing code for electron cyclotron heating & current drive calculations in tokamaks.
Torray-FOM [36]	EC/waves	Egbert Westerhof	Torray-FOM is a ray-tracing code for electron cyclotron heating & current drive calculations in tokamaks.
bbnbi [37]	NBI/source	Otto Asunta	Calculate the deposition rates of neutrals beam particles, i.e. the input source for Fokker-Planck solvers (not the heating and current drive). Note that the number of markers generated by BBNBI is described by the kepler variable `number_nbi_markers_in` .
nemo [38]	NBI/source	Mireille Schneider	Calculate the deposition rates of neutrals beam particles, i.e. the input source for Fokker-Planck solvers (not the heating and current drive). Code-parameter documentation can be found here [39].
nuclearsim [40]	nuclear/source	Thomas Johnson	Simple code for nuclear sources from thermal/thermal reactions. Code-parameter documentation can be found here [41].
nbisim [42]	NBI, alphas / Fokker-Planck	Thomas Johnson	Simple Fokker-Planck code calculating the collisional ion and electron heating from a particle source, either NBI or nuclear. Code-parameter documentation can be found here [43].
risk [44]	NBI Fokker-Planck	Mireille Schneider	Bounce averaged steady-state Fokker-Planck solver calculating the collisional ion and electron heating from a particle source and the NBI current drive. Code-parameter documentation can be found here [45].
spot [46]	NBI, alphas and ICRF Fokker-Planck	Mireille Schneider	Monte Carlo solver for the Fokker-Planck equation. Traces guiding centre orbits in a steady state magnetic equilibrium under the influence of Coloumb collisions and interactions with ICRF waves (through the RFOF library). The code can also be used for NBI and alpha particle modelling as it can handle source terms from the distsource CPO.
ascot4serial [47]	NBI, alphas, ICRF / Fokker-Planck	Otto Asunta/Seppo Sipila	Monte Carlo Fokker-Planck solver calculating the collisional ion and electron heating from a particle source and the NBI current drive.

[33] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_gray
[34] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_lion.html
[35] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_travis
[36] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_travis
[37] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_bbnbi
[38] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_bbnbi
[39] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_nemo.html
[40] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_nuclearsim
[41] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_nuclearsim.html
[42] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_nbisim
[43] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_nbisim.html
[44] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_risk
[45] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_risk.html
[46] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_spot
[47] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_ascot

Code name	Code Category	Contact persons	Short description
ascot4parallel [48]	NBI, alphas, ICRF / Fokker-Planck	Otto Asunta/Seppo Sipila	Monte Carlo Fokker-Planck solver calculating the collisional ion and electron heating from a particle source and the NBI current drive.
Lion	IC / waves	Olivier Sauter and Laurent Villard	Global ICRF wave solver. Code-parameter documentation can be found here [49].
Cyrano	IC / waves	Ernesto Lerche and Dirk Van Eester	Global ICRF wave solver. Code-parameter documentation can be found here [50].
Eve (not yet in ETS)	IC / waves	Remi Dumont	Global ICRF wave solver
StixReDist	IC / waves	Dirk Van Eester and Ernesto Lerche	1d Fokker-Planck solver for ICRF heating.
ICdep	IC / waves	Thomas Johnson	Generates Waves-cpo with an IC wave field with Gaussian deposition profiles described by a combination of antenna-cpo input and through code parameters input. Code-parameter documentation can be found here [51].
ICcoup [52]	IC / coupling	Thomas Johnson	Simple model for the coupling waves from ion cyclotron antennas to the plasma. Code-parameter documentation can be found here [53].

2.1.3.5 Events actors

Code name	Code Category	Contact persons	Short description
pelletactor	pellet	Denis Kalupin	
pellettrigger	pellet	Denis Kalupin	
sawcrash_slice	sawteeth	Olivier Sauter	
sawcrit	sawteeth	Olivier Sauter	
runaway_indicator [54]	runaway	Roland Lohner och Gergo Pokol	Indicating the presence of runaway electrons: 1) Indicate, whether electric field is below the critical level, thus runaway generation is impossible. 2) Indicate, whether runaway electron growth rate exceeds a preset limit. This calculation takes only the Dreicer runaway generation method in account and assumes a velocity distribution close to Maxwellian, therefore this result should be considered with caution. The growth rate limit can be set via an input of the actor. Limit value is set to 10^12 particle per second by default. (This growth rate generates a runaway current of approximately 1 kA considering a 10 seconds long discharge.)

2.1.3.6 Non-physics actors

The ETS uses the following list of non-physics actors:
`addECant, addICant, backgroundtransport, calculateRHO, changeocc, changepsi, changeradii, checkconverge controlAMIX, coredelta2coreprof, correctcurrent, deltacombiner, emptydistribution, emptydistsource, emptywaves, eqinput, etsstart, fillcoreimpur, fillcoreneutrals, fillcoreprof, fillcoresource, fillcoretransp, fillequilibrium, fillneoclassic, filltoroidfield, gausiansources, geomfromcpo, hcd2coresource, ignoredelta, ignoreimpurity, ignoreneoclassic, ignoreneutrals, ignorepellet, ignoresour ignoretransport, IMP4dv, IMP4imp, importimptransport, itmimpurity, itmneutrals, merger4distribution, merger4distsource, merger4waves, nbifiller, neoclassic2coresource, neoclassic2coretransp, parabolicprof plasmacomposition, PowerFromArray, PowerModulation, profilesdatabase, readjustprof, sawupdate_slice, scaleprof, sourcecombiner, sourcedatabase, transportcombiner, transportdatabase, wallFiller and waves2sources.`

2.2 ETS_A 4.10a

The development on this version of the workflow discontinues. You can use it for production runs using 4.10a data structure and UAL. New functionalities shall be requested to 4.10b version!

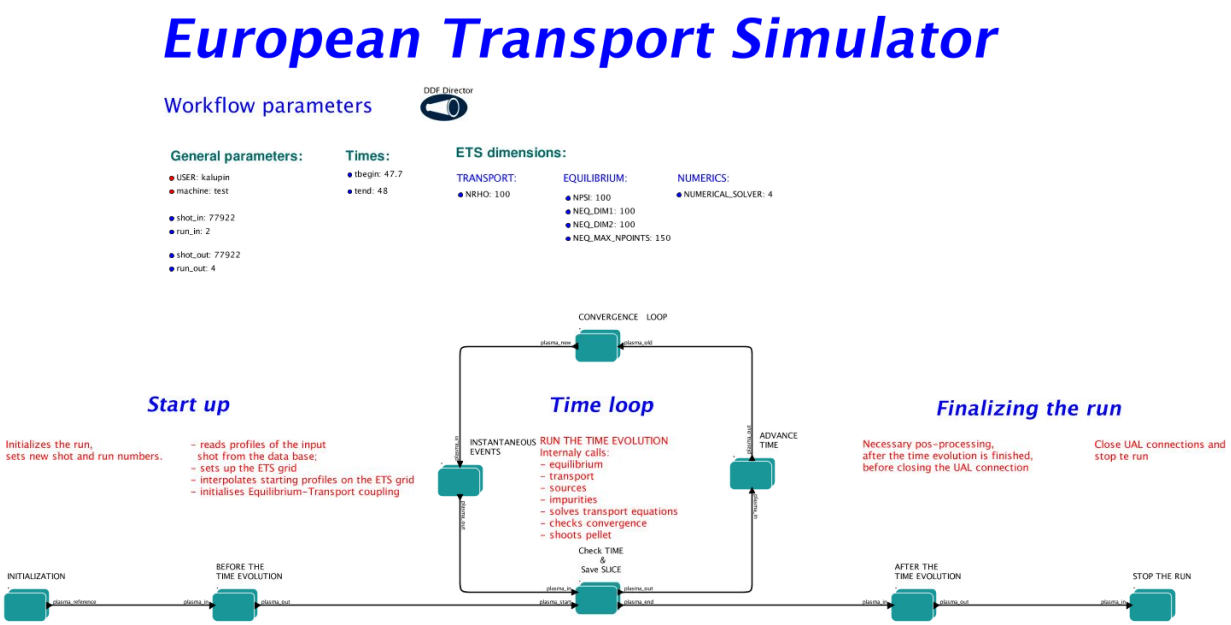
[48] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_ascot
[49] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_lion.html
[50] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_cyrano.html
[51] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_icdep.html
[52] https://www.efda-itm.eu/ITM/html/imp5_listcodes.html#imp5_listcodes_iccoup
[53] https://www.efda-itm.eu/ITM/html/imp5_code_parameter_documentation_iccoup.html
[54] http://portal.efda-itm.eu/twiki/bin/view/Main/HCD-codes-runin-usermanual

ETS_A workflow in KEPLER :

- uses as actors and composite actors from other IMPs, thus for the most recent versions of them please check with relevant project
- complex, but clearly structured workflow, which offers user friendly interface for configuring the simulation
- allows for easy modifications (*connecting new modules, or reconnecting the parts of the workflow*) through the easy graphical interface
- provides users with all updates through the version control system
- still actively developing tool

The list and status of available physics models for the ETS_A can be found here [55].

Contact person: ***denis.kalupin@euro-fusion.org?subject=ETS%20in%20KEPLER*** ***(Skype: dkalupin)***



2.2.1 Obtaining the ETS

Copy the ETS workflow to your space:

```
>svn co https://gforge6.eufus.eu/svn/keplerworkflows/trunk/4.10a/imp3/ets $ITMSCRATCH/ETS_WORKFLOWS
```

Compile ETS actors:

```
>cd $ITMSCRATCH/ETS_WORKFLOWS
>make import_ets
```

[55] https://www.eufus.eu/documentation/ITM/html/ets_status.html

2.2.2 Updating the ETS

If you have already a copy of the ETS you do not need to check it out again!!!
If you like to update everything (WORKFLOW + ACTORS + VISUALIZATION + INPUT DATA)

```
>cd $ITMSCRATCH/ETS_WORKFLOWS
>svn update
>make import_ets
```

To update ETS actors go inside your ETS_ACTORS:

```
>cd $ITMSCRATCH/ETS_WORKFLOWS
>svn update
>make import_actors
```

To update the workflow go inside your ETS_WORKFLOWS:

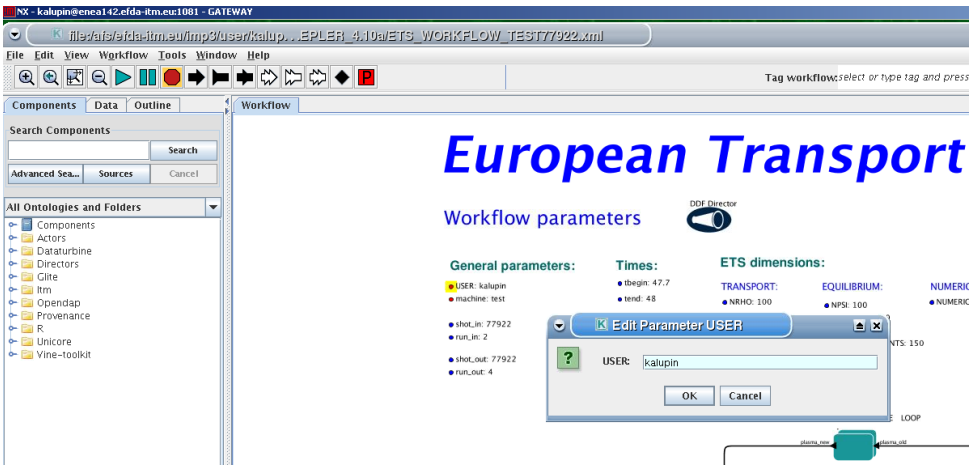
```
>cd $ITMSCRATCH/ETS_WORKFLOWS
>svn update
```

To update visualization scripts go inside your $KEPLER/kplots:

```
>svn update
```

This is ALL you need to do for updates!

2.2.3 Executing the ETS



Open ETS workflow in Kepler:

```
>kepler.sh $ITMSCRATCH/ETS_WORKFLOWS/ETS_WORKFLOW.xml
```

- on the top of the workflow, change the parameter "user" to your user_ID.
You can run the workflow!!!

2.2.4 Configuring the ETS run

2.2.4.1 WORKFLOW PARAMETERS

2.2.4.1.1 General Parameters

- USER - your userid
- MACHINE - machine name (database name) for which comutations are done
- SHOT_IN - input shot number
- RUN_IN - input run number
- SHOT_OUT - output shot number
- RUN_OUT - output run number
- NUMERICAL_SOLVER - choice of the numerics solving transport equations (RECOMENDED SELECTION: 3 or 4)

2.2.4.1.2 Space resolution

- NRHO - number of radial points for transport equations
- NPSI - number of points for equilibrium 1-D arrays
- NEQ_DIM1 - number of points for equilibrium 2-D arrays, first index
- NEQ_DIM2 - number of points for equilibrium 2-D arrays, second index
- NEQ_MAX_NPOINTS - maximum number of points for equilibrium boundary

2.2.4.1.3 Time resolution

Start and End time

- TBEGIN - Computations start time
- TEND - Computattions end time

Time spep

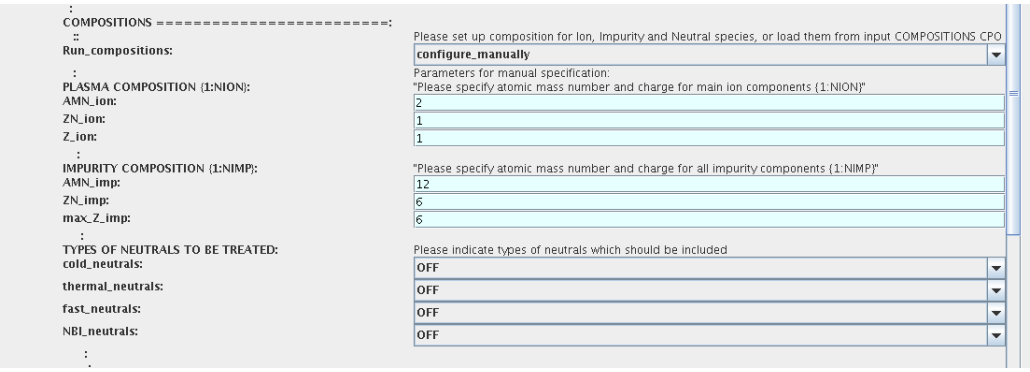
- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor
- TAU:specify value of the time step in [s]
- TAU_OUT: specify value of the output time interval in [s]
- Commit

2.2.4.2 PLASMA, IMPURITY and NEUTRALS COMPOSITION

Before starting the run you need to define types of main and impurity ions and types of neutrals to be included in simulations.

To set up the composition:

- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor
- choose one of modes for setting "Run_compositions"
  "from_input_CPO" - will pick up the COMPOSITIONS structure of the COREPROF CPO from the input shot;
  "configure_manually" - will force the composition from the values specified below
- specify values of AMN_ion, ZN_ion and Z_ion for ions, from the first ion to the last [1:NION], separated by commas
- specify values of AMN_imp, ZN_imp and max_Z_imp for impurity ions, from the first to the last [1:NIMP], separated by commas
- choose the neutrals types, which should be switched "ON"
- Commit



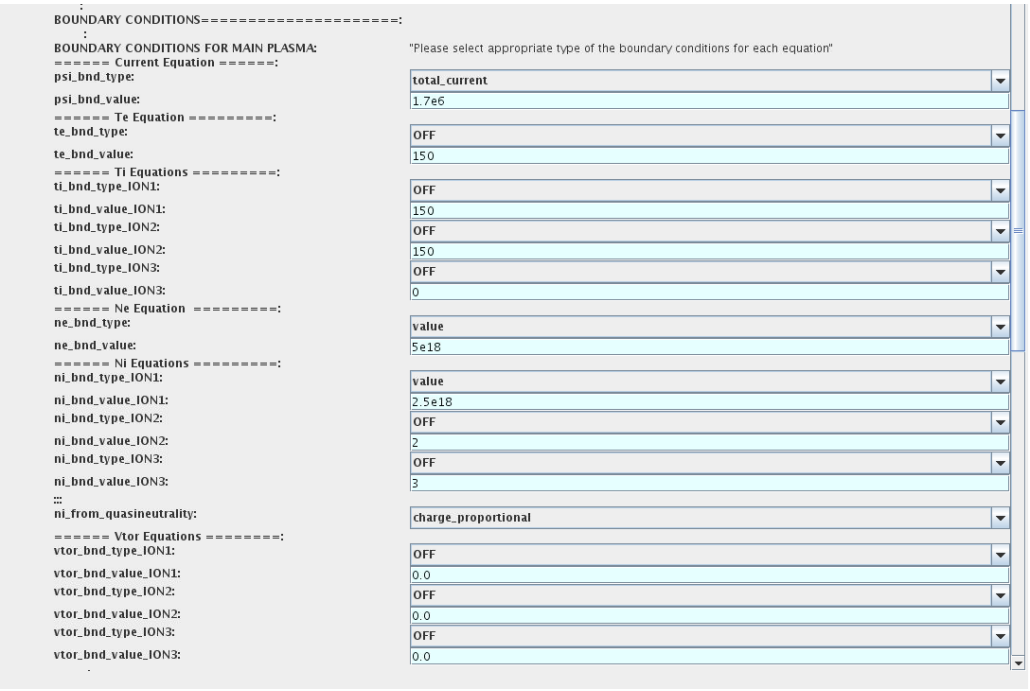
2.2.4.3 EQUATIONS TO BE SOLVED AND BOUNDARY CONDITIONS

2.2.4.3.1 MAIN PLASMA

Before starting the run you need to select the type and value of the boundary conditions for all equations. Please note that the value should correspond to the type. All equations allow for following types of boundary conditions:

- *OFF* - equation is not solved, initial profiles will be kept for whole run
- *value* - edge value should be specified
- *gradient* - edge gradient should be specified
- *scale_length* - edge scale length should be specified
- *generic* - 3 coefficients (a1,a2,a3) should be provided: a1*y + a2*y = a3
- *value_from_input_CPO* - equation is solved, edge value evolution will be red from input shot
- *profile_from_input_CPO* - equation is not solved, profile evolution will be red from input shot

The particular equation will be activated if the boundary condition type for it is other than *OFF* !



To set up boundary conditions:

- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor
- select appropriate boundary condition for each equation
- specify values for boundary conditions corresponding to the type and to the ion component
- Commit

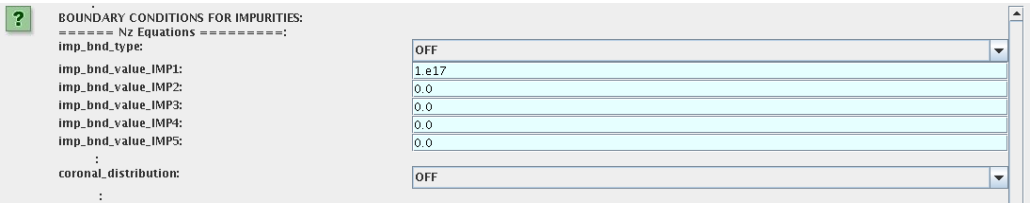
!!! If electron density is solved, all ions with ni_bnd_type=OFF will be computed from the quasineutrality condition and scaled proportional to specified *ni_bnd_value* or inversely proportional to their charge *(charge_proportional)* . This is defined by option: *ni_from_quasineutrality* .

2.2.4.3.2 IMPURITY

You can set up the boundary conditions for impurity ions in a similar way as for main ions.
!!! Note, that at the moment only types: *OFF* ; *value* and *value_from_input_CPO* are accepter by impurity solver.

To set up boundary conditions:

- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor
- select appropriate boundary condition for each impurity species (OFF-equation is not solved)
- specify values for boundary density of each impurity component [1:MAX_Z_IMP], separated by commas
- Commit



Interface for impurity boundary condition has additional option , *coronal_distribution* , that allow to preset the edge values or entire profiles of individual ionization states from coronal distribution. In tis case only single value is required to be specified for each impurity boundary value.
The options are:

- *OFF* - the boundary values for impurity densities will be as they are specified above;
- *boundary_conditions* - the boundary densities will be renormalized with corona, using the first element from above as a total density
- *boundary_conditions_and_profiles* - the boundary densities and starting profiles will be renormalized with corona, using the first element from above as a total density

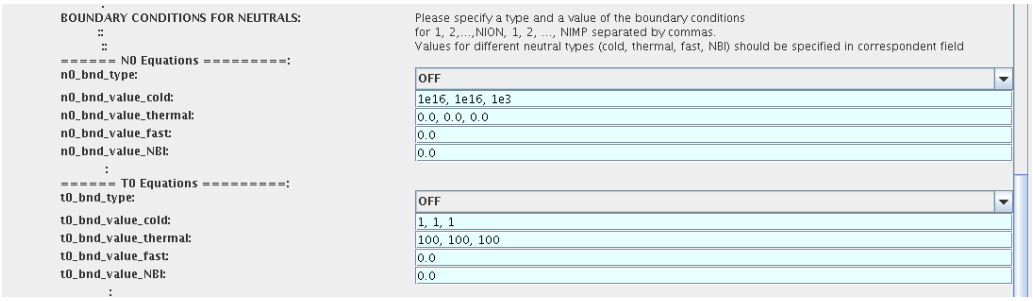
2.2.4.3.3 NEUTRALS

!!! AT THE MOMENT BOUNDARY CONDITIONS FOR NEUTRAL VELOCITIES ARE DISABLED, MIGHT BE ADDED ON REQUEST

Note, that ALL values should be specified in the order: {*1, 2, 3 ...NION, 1, 2, 3, ...NIMP*}

To set up boundary conditions:

- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor
- select appropriate boundary condition for each neutral species (OFF-equation is not solved)
- specify values for boundary density and temperature of each neutral component [1, 2, 3 ...NION, 1, 2, 3, ...NIMP], separated by commas
- Commit



2.2.4.3.4 INPUT PROFILES INTERPOLATION

You are going to start the ETS run from some input shot, which might contain some conflicting rho grids. Thus there is a choice for the user to decide on the grid on which the starting profiles should be load by the worflow, *Interpolation_of_input_profiles* .

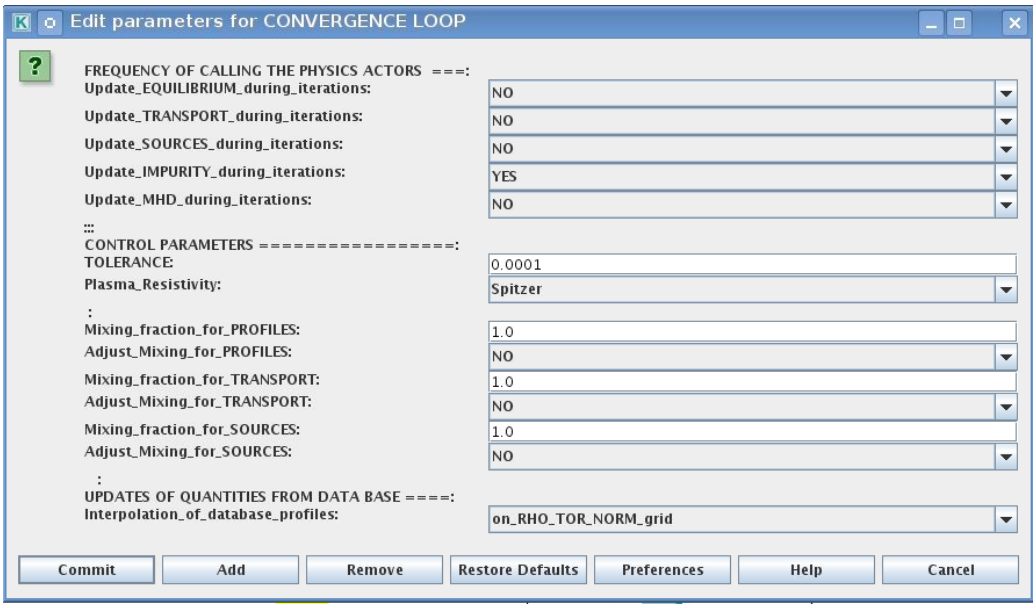
To define the interpolation grid select:

- *on_RHO_TOR_grid* - interpolate input profiles based on the grid specyfied in [m];
- *on_RHO_TOR_NORM_grid* - interpolate input profiles based on normalised rho grid [0:1]

2.2.4.4 CONVERGENCE LOOP

ETS updates input from different physics actors in a sequence, which is finished by solving the transport equations. Ther are possible none-linear couplings between different parts of the system. These nonelinearities are trited by the ETS using iterations.
The decision to step in time is made by the ETS based on the criteria that the maximum relative deviation of main plasma profiles is lower than some predefined tolerance.
There is a number of settings and sitches in the ETS that are used by the iterative scheme. To edit them do:

- right click on the box CONVERGENCE LOOP
- select Configure actor to edit settings
- choose your settings
- Commit



Switches in the field *FREQUENCY OF CALLING THE PHYSICS ACTORS* define how many times the the actors of a certain cathegory (equilibrium, transport, etc.) should be called in a single time step.
By selecting *YES* all actors of this cathegory will be called every iteration
By selecting *NO* all actors of this cathegory will be called only ones in a time step

Switches and parameters in the field *CONTROL PARAMETERS* define how iterations are done

- *Tolerance* - defines the maximum relative error of profiles change compared to previous iteration. If it is achieved the time steping is done.

For highly none-linear case the required precision can be achieved faster by the iterative scheme if only fraction of the new solution is mixed to the previous state.
The following scheme is adopted by the ets to reduce none-linearities in profiles, transport coefficients and sources:

```
Y = (Amix * Y+) + ((1-Amix)*Y-)
```

where Amix is the mixing fraction

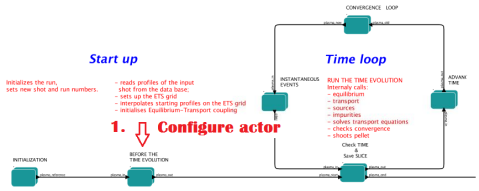
You can activate the mixing of profiles, transport coefficient and sources by selecting the corresponding *Mixing_fraction_...* to be between [0:1]
You also can activate the authomatic ajustment of this fraction by selecting: *Ajust_Mixing_for_...* to *YES*

2.2.4.5 EQUILIBRIUM

2.2.4.5.1 Starting Settings

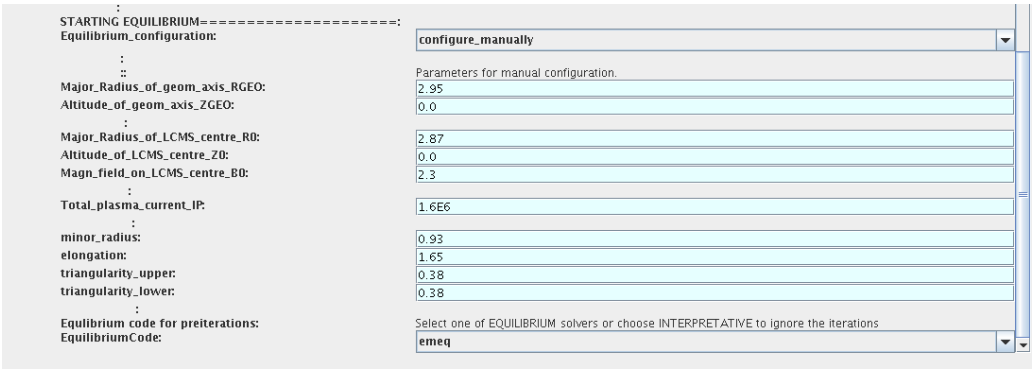
Before starting the run you need to set up your initial equlibrium. There are several options to do it: if your input shot contains the consistent equilibrium with all necessary parameters - you can start immediately from it; if your input shot contains the equilibrium but it is not consistent or some parameters are missing you can check it automatically; if your input equilibrium is corrupt or not present - you can define the starting equlinbrium by tree moment description. To select your starting equilibrium please do:

- right click on the box BEFORE THE TIME EVOLUTION
- select Configure actor to edit settings
- Select your settings or specify values
- Commit



SETTINGS:

- *Equilibrium_configuration* - select *configure_manually* if you like to specify configuration below; select *from_input_CPO* if all quantities should be picked up from the input CPO
- *Major_Radius_of_geom_axis_RGEO* - radius of the geometrical centre of the vessel [m]
- *Altitude_of_geom_axis_ZGEO* - altitude of the geometrical centre of the vessel [m]
- *Major_Radius_of_LCMS_centre_R0* - radius of the plasma centre [m]
- *Altitude_of_LCMS_centre_Z0* - altitude of the plasma centre [m]
- *Magn_field_on_LCMS_centre_B0* - vacume magnetic field at R0 [T]
- *Total_plasma_current_IP* - plasma current within the LCMS [A]
- *Minor_radius* - minor radius of the LCMS [m]
- *Elongation* - elongation of the LCMS [-]
- *Triangularity_upper* - upper triangularity of the LCMS [-]
- *Triangularity_lower* - lower triangularity of the LCMS [-]
- *Equilibrium code* - select one of available equilibrium solvers to check the consistency between starting equilibrium and current profile; use *INTERPRETATIVE* if you trust your input data (in this case the check will be ignorred).

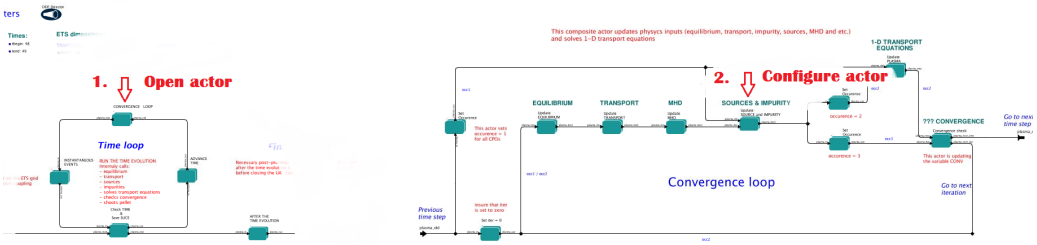


Please note, that different equilibrium solvers might require slightly different input. Thus it is a user responsibility to check that the information inside input shot/run is enough to run selected equilibrium solver.

2.2.4.5.2 Run Settings

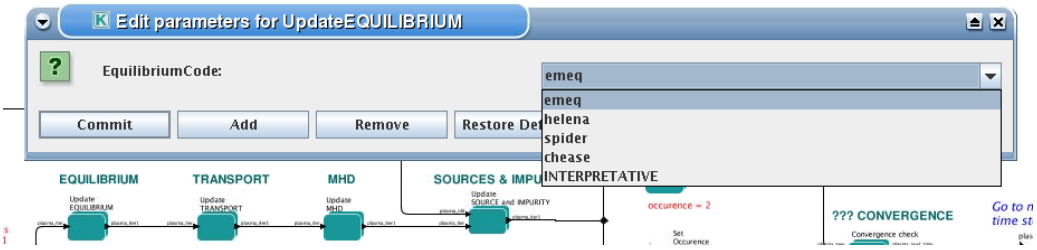
There are several equilibrium solvers connected to the ETS. You can select the one of them.Therefore please do:

- right click on the box CONVERGENCE LOOP
- select Open actor
- right click on the box EQUILIBRIUM
- select Configure actor to edit settings
- choose your equilibrium solver
- Commit



INTERPRETATIVE means that the ETS will not update the equilibrium, instead it will be using the initial equilibrium.

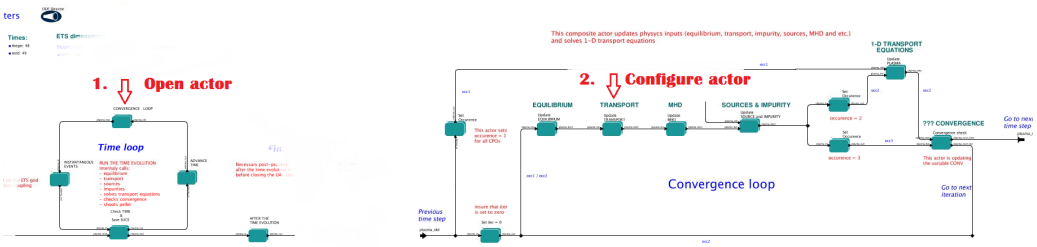
Please note, that it is better to select the same code as you used for pre-iterrations. Because outputs of different equilibrium solver are not necessary done with the same resolution. Therefore the routine saving the information to the data base might brake due to uncompatible sizes of some signals.



2.2.4.6 TRANSPORT

The settings for TRANSPORT can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

- right click on the box CONVERGENCE LOOP
- select Open actor
- right click on the box TRANSPORT
- select Configure actor to edit settings
- choose your settings
- Commit



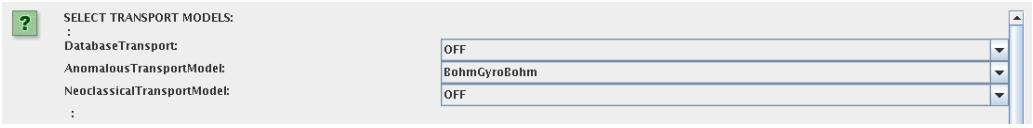
2.2.4.6.1 Choice of transport model

ETS constructs the total transport coefficients from the combination of Anomalous transport (model choice), Neoclassical transport (model choice) and Database transport (transport coefficients be saved to the input shot)

```
D_tot = D_DB*M_DB + D_AN*M_AN + D_NC*M_NC
```

You should choose from the list of available models in each cathegory or switch it OFF

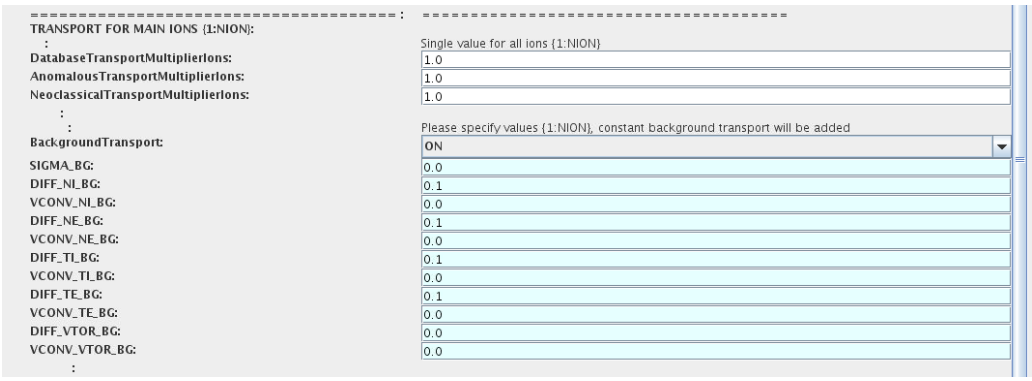
The list of available transport models can be found here [56].



2.2.4.6.2 Main plasma transport

In this section you define how total transport coefficients for main ions should be constructed from contributions provided by different models. You need to provide the multipliers for Anomalous, Neoclassical and Database contributions, which will determine their weights in total transport coefficient.

You also can add the constant background level for each coefficient (ion coefficients are expected to be the string {1:NION}, separated by commas)



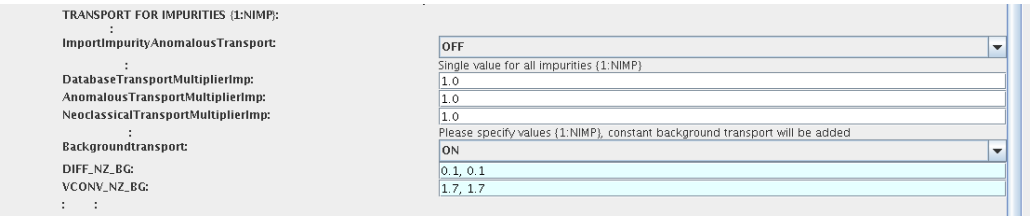
2.2.4.6.3 Impurity transport

In this section you define how total transport coefficients for impurity ions should be constructed from contributions provided by different models. You need to provide the multipliers for Anomalous, Neoclassical and Database contributions, which will determine their weights in total transport coefficient.

You also can add the constant background level for each coefficient (coefficients are expected to be the string {1:NIMP}, separated by commas)

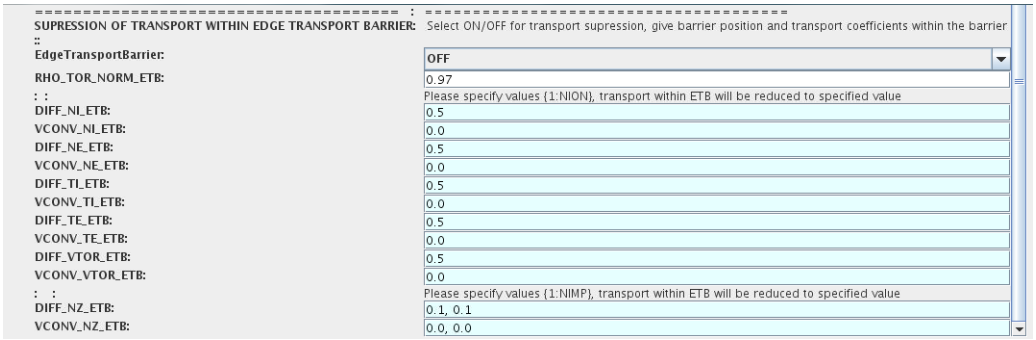
In addition, there is an option to import the Anomalous component of transport coefficient *from_first_ion* or *from_electrons* (the same anomalous contribution will be added to all impurity components, all ionization states)

[56] https://www.eufus.eu/documentation/ITM/html/ets_status.html



2.2.4.6.4 Edge transport barrier

In this section you can artificially supress the transport outside of specified *RHO_TOR_NORM_ETB* . Total transport coefficients for all transport channels (ne, ni, nz, Te, Ti,...) will be reduced to constant values specified below (ion and impurity coefficients are expected to be the strings {1:NION}) and {1:NIMP} respectively)



2.2.4.6.5 Total transport coefficients

Profiles of the total transport coefficient for each channel are obtained from the the individual contributions (Data Base, Anomalous, Neoclassical and Background) as a summ of all activated transport models multiplied with coefficients specified on the interface of the composite actor.

```
X_tot = X_DB*DBM + X_AN*ANM + X_NC*NCM + X_BG*BGM
```

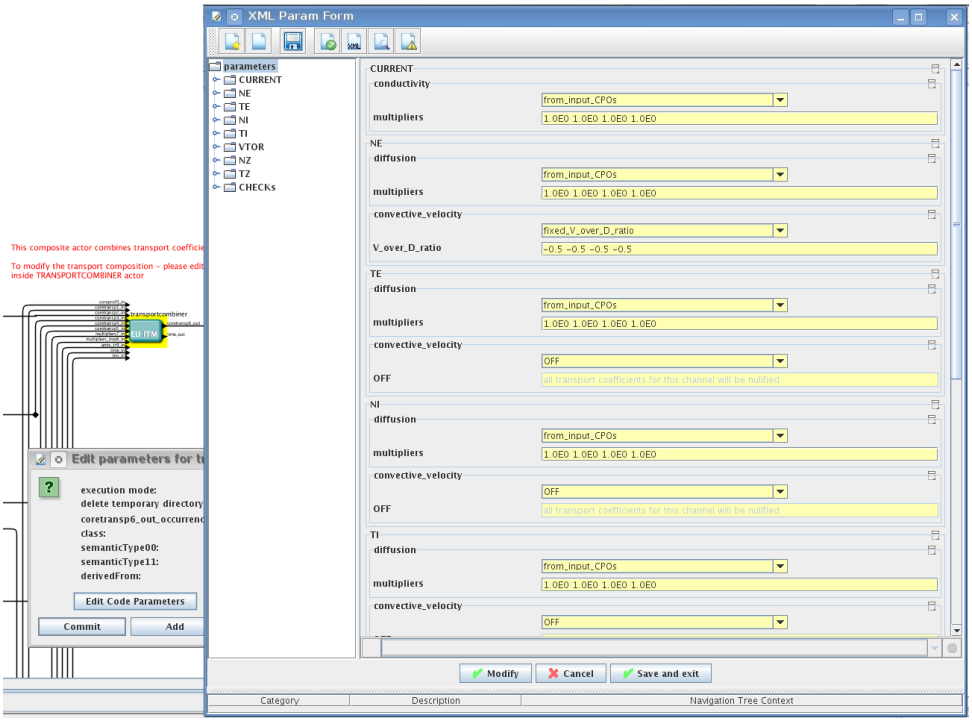
!!! Note, that contributions to all transport equations will be multiplied with the same value.
For example: if AnomalousTransportMultiplier=3.0, then contibutions from selected anomalous transport model to each transport equation will be multiplied with 3.0

The fine tuning of of transport coefficients can be done through editing the XML code parameters of the transport combiner actor:

- right click on the box TRANSPORT
- select Open actor to edit settings
- right click on the box Transport Combiner
- select Open actor to edit settings
- right click on the box transportcombiner
- select Configure actor
- click Edit Code Parameters
- If you select ***OFF*** contributions from all transport models to this channel will be nullified; If you select ***from_input_CPOs*** the transport channel will be activated, and the total transport coefficient will be combined from active tranport models; For convective velocity there is an additional option ***fixed_V_over_D_ratio*** , by selecting this the combiner will ignore the convective components provided by transport nmodels. The convective velocity will be determined from the total diffusion coefficient by applying fixed V/D ratio (*for inward pinch the values should be negative!*).
  For all active channels you can adjust multipliers for combining contributions from different transport models (array of four space separated values is expected):

- first position - Data Base transport coefficients;
- second position Anomalous transport coefficients;
- third position Neoclassical transport coefficients;
- fourth position Background (constant level) transport coefficients;

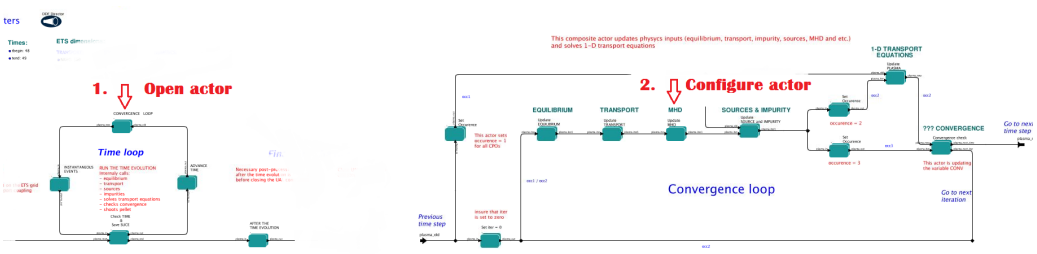
- save and exit
- Commit



2.2.4.7 MHD

The settings for MHD type of events can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

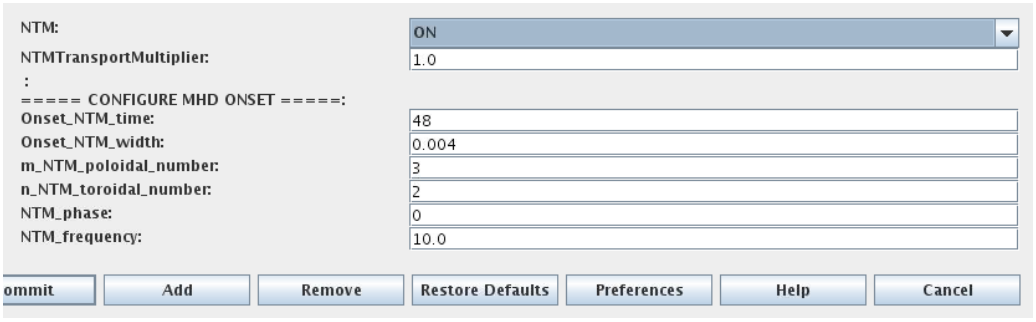
- right click on the box CONVERGENCE LOOP
- select Open actor
- right click on the box MHD
- select Configure actor to edit settings
- choose your settings
- Commit



At the moment ETS allows only for NTM to be activated. The sawtooth module is expected to be deployed before ITM Code Camp in Slovenia.

User can ajust the following NTM settings:

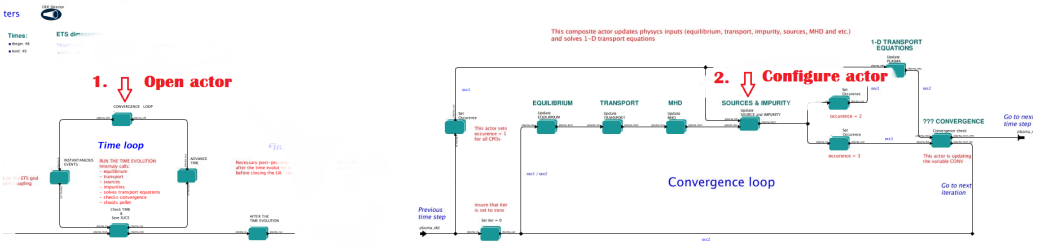
- NTM *ON* means that ETS will add the NTM driven transport to the total transport coefficient; *OFF* -ignored
- NTMTransportMultiplier the transport contrinution from NTM will be multiplied with this value
- Onset_NTM_time - activation time for the NTM mode
- Onset_NTM_width - starting width of the mode
- m_NTM_poloidal_number
- n_NTM_toroidal_number
- NTM_phase
- NTM_frequency



2.2.4.8 SOURCES AND IMPURITY

The settings for SOURCES AND IMPURITY can be done inside the CONVERGENCE LOOP composite actor. Therefore please do:

- right click on the box CONVERGENCE LOOP
- select Open actor
- right click on the box SOURCES AND IMPURITY
- select Configure actor to edit settings
- choose your settings
- Commit

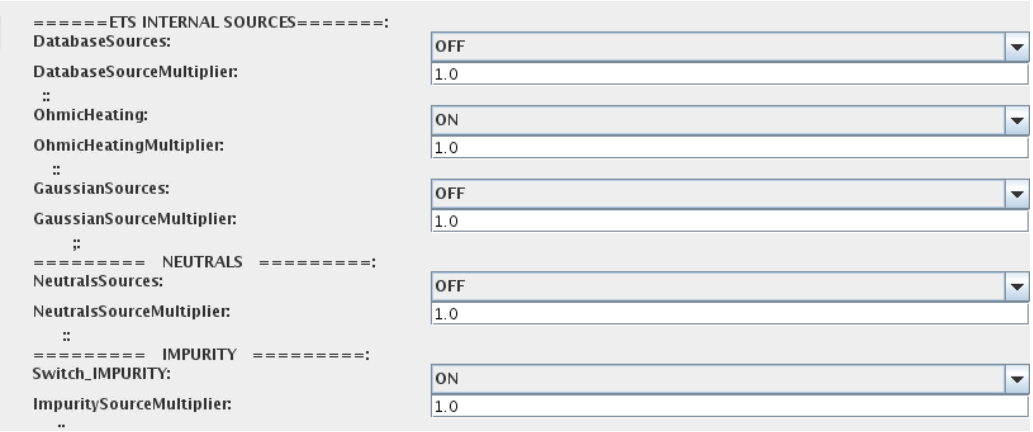


2.2.4.8.1 IMP3 sources

There is a number of sources developed by IMP3 project, which are actors or internal routines of the transport solver. You can activate them by selecting *ON / OFF* in front of corresponding source:

- Database Sources *ON* - ETS will pick up the evolution of source profiles saved to your input shot/run; *OFF* -ignored
- Ohmic Heating *ON* - ETS will compute Ohmic heating internaly; *OFF* -ignored

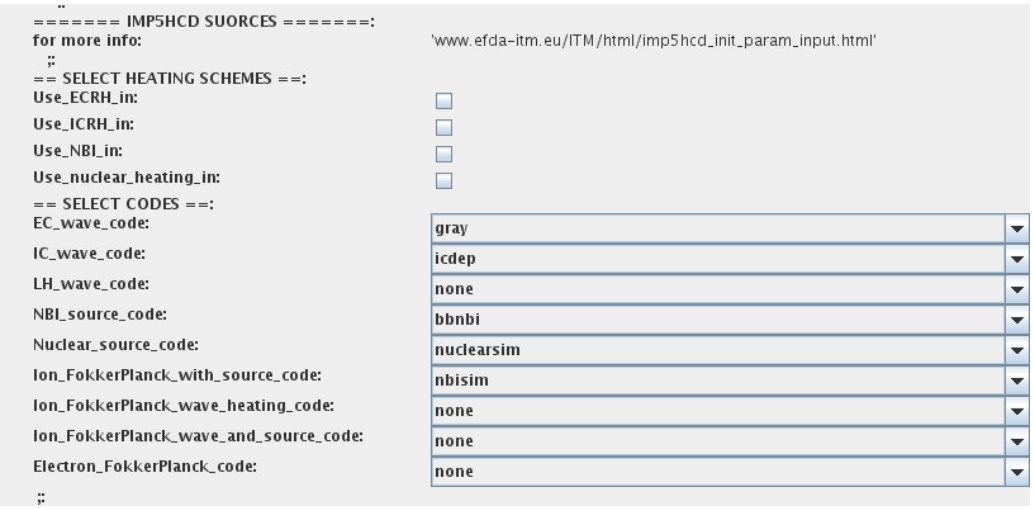
- Gaussian Sources *ON* - ETS will add sources from the Gaussian source actor (you can configure heat and particle deposition profiles by editing the code parameters of the actor); *OFF* -ignored
- Neutral Sources *ON* - Fluid neutrals will be solved according to the boundary conditions specified on Before_time_evolution composite actor interface; *OFF* -ignored
- Switch_IMPURITY *ON* - Impurity density and radiative sources will be computed; *OFF* -ignored; *INTERPRETATIVE* profiles of impurity density will be read from input shot/run



2.2.4.8.2 IMP5HCD sources

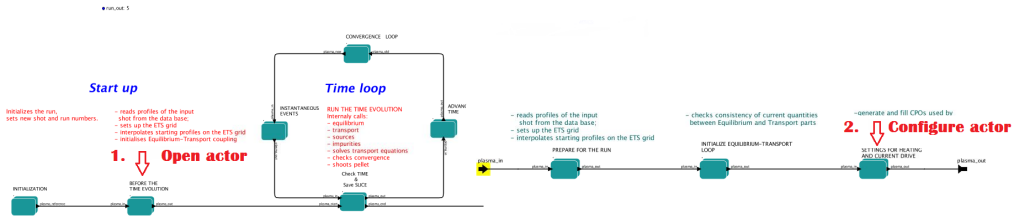
There is a number of sources developed by IMP5 project, that are incorporated by the ETS workflow.

For the IMP5HCD sources please activate the type of heating source, by ticking the box in front of it, and select the code to simulate it.



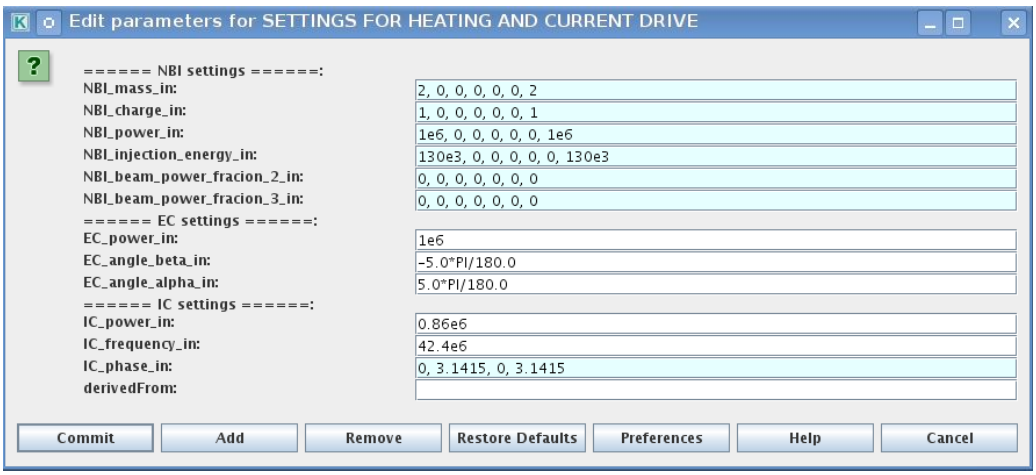
You also need to configure initial IMP5HCD settings. Therefore please:

- right click on the box BEFORE THE TIME EVOLUTION
- select Open Actor
- right click on the box SETTINGS FOR HEATING AND CURRENT DRIVE
- select Configure actor
- edit the stettings
- Commit



The detailed information on initial IMP5HCD settings can be found here [57].

Please note that settings for NBI are done independent for each PINI. Therefore, for NBI settings, please insert the values separated by commas. The number of the element in the array corresponds to the number of activated PINI. Maximum accepted number of PINIs = 16.



2.2.4.8.3 Power control

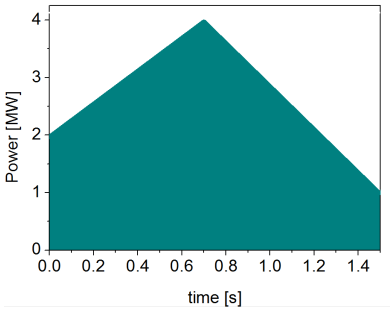
You also can activate the power control for the IMP5HCD sources.

[57] https://www.eufus.eu/documentation/ITM/html/imp5_imp5hcd.html

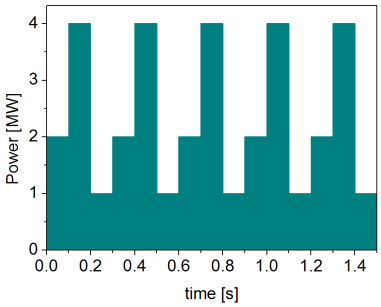
```
!
== POWER CONTROL ==:
!!
NBI_power_control:        specific
Times_NBI:                0, 100
Power_NBI_P1:             2E6, 2E6
Power_NBI_P2:             0
Power_NBI_P3:             0
Power_NBI_P4:             0
Power_NBI_P5:             0
Power_NBI_P6:             0
Power_NBI_P7:             2E6, 2E6
Power_NBI_P8:             0
Power_NBI_P9:             0
Power_NBI_P10:            0
Power_NBI_P11:            0
Power_NBI_P12:            0
Power_NBI_P13:            0
Power_NBI_P14:            0
Power_NBI_P15:            0
Power_NBI_P16:            0
tstart_NBI_control:       48
tend_NBI_control:         49
frequency_NBI_control:    100
  :
ECRH_power_control:       OFF
Times_ECRH:               48
Power_ECRH:               3e6,0e6,2e6
tstart_ECRH_control:      48
tend_ECRH_control:        49
frequency_ECRH_control:   100
  :
ICRH_power_control:       OFF
Times_ICRH:               0
Power_ICRH:               0
tstart_ICRH_control:      47
tend_ICRH_control:        55
frequency_ICRH_control:   100
```

If the POWER_CONTROL is not *OFF* , there are two modes of operation: *specific* and *frequency*

For *specific* you should specify the time sequence separated by commas and the corresponding power sequence (where first power level corresponds to the first time, second to second and etc.). Linear interpolation will be done between the sequence points.
For example: if you give the power sequence = 2e6,4e6,1e6 and times = 0.0, 0.7, 1.5 (s) the delivered power would be:



For *frequency* you should specify the power levels sequence separated by commas, start and end time of the power control and the frequency of switching between these levels.
For example: if you give the power sequence = 2e6,4e6,1e6 and frequency = 10 (Hz) tstart =0.0 (s) tend = 1.5 (s) the delivered power would be:



2.2.4.8.4 Total power

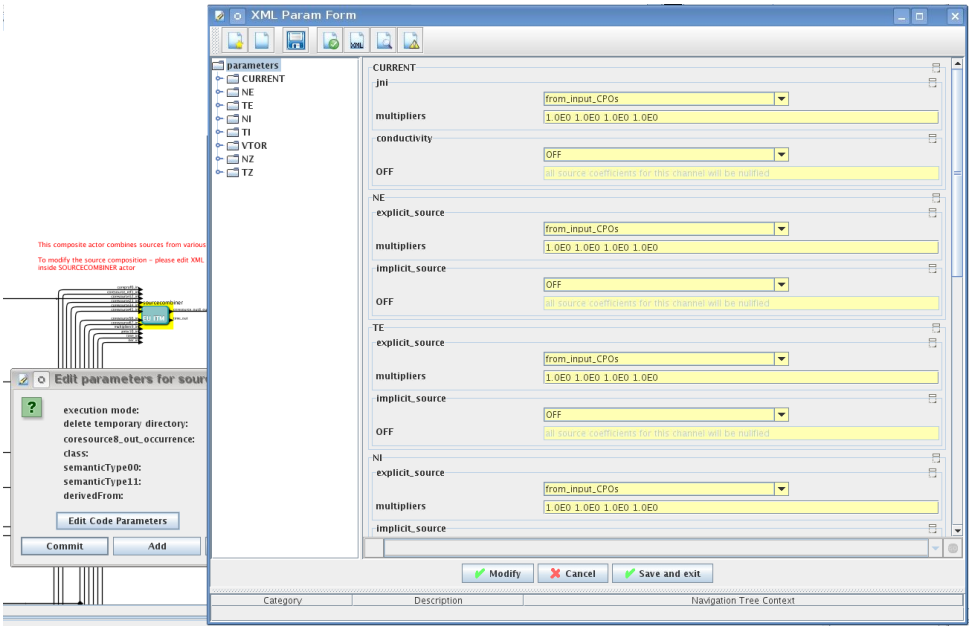
Profiles of the total source for each channel are obtained from the the individual contributions (Data Base, Gaussian, Neutrals, Impurity and HCD) as a summ of all activated sources multiplied with coefficients specified on the interface of the composite actor.

```
S_tot = S_DS*DSM + S_GS*GSM + S_Neu*NeuSM + S_IMP*IMPSM + S_HCD*HCDSM
```

!!! Note, that contributions to all transport equations will be multiplied with the same value.
For example: if ImpuritySourceMultiplier=3.0, then contibutions from impurity to Se, Sz and Qe will be multiplied with 3.0

The fine tuning of of sources can be done through editing the XML code parameters of the source combiner actor:

- right click on the box SOURCES and IMPURITY
- select Open actor to edit settings
- right click on the box Source Combiner
- select Open actor to edit settings
- right click on the box sourcecombiner
- select Configure actor
- click Edit Code Parameters
- If you like the sources to the particular equation being activated - select ***from_input_CPOs*** ; if you select ***OFF*** contributions from all sources to this channel will be nullified.
  For active channels you can adjust multipliers for combining contributions from different source modules (array of five space separated values is expected):
  - first position - Data Base sources;
  - second position Gaussian sources;
  - third position HCD sources;
  - fourth position Neutral sources;
  - fifth position Impurity sources.
- save and exit
- Commit



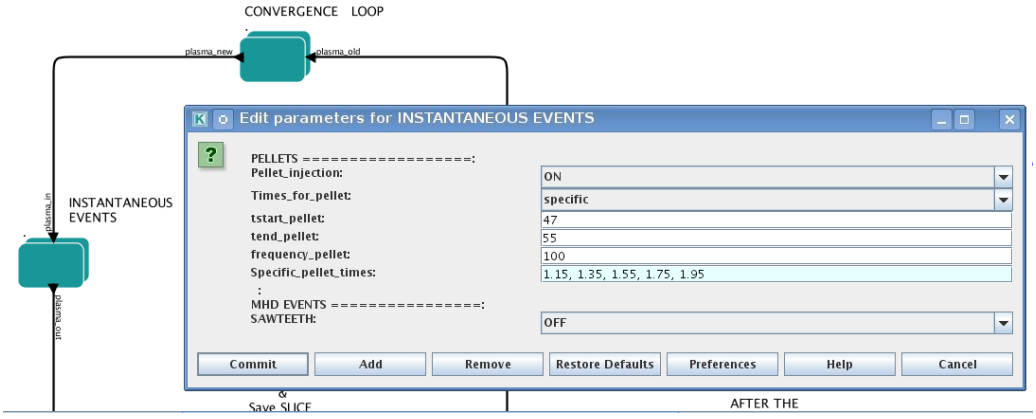
2.2.4.9 INSTANTANEOUS EVENTS

At the moment, user can swith ON and OFF two types of events: PELLET and SAWTOOTH

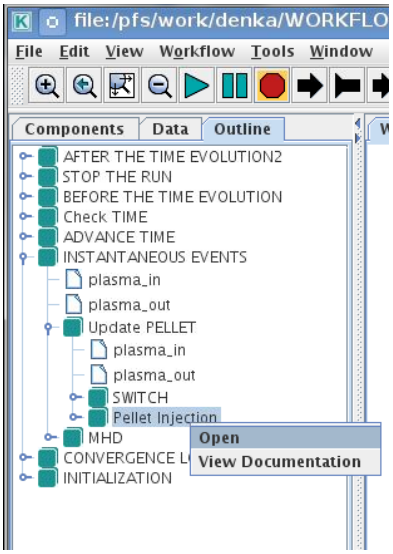
2.2.4.9.1 PELLET

At the top level of the workflow you can configure times for pellet injection

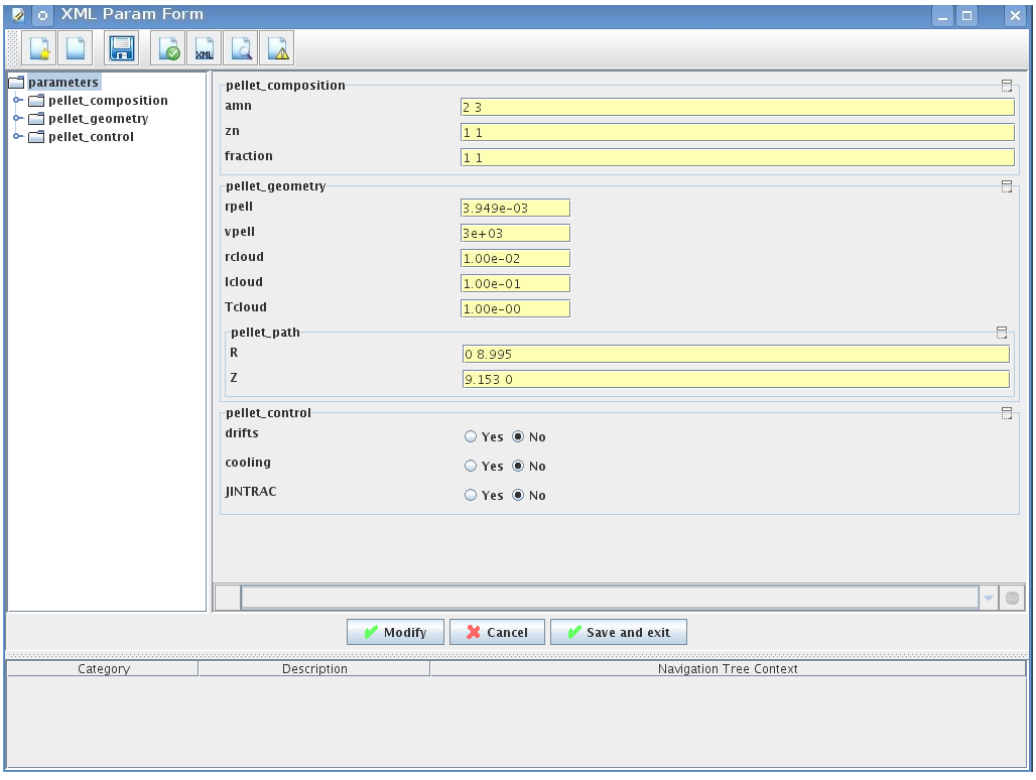
- right click on the box INSTANTANEOUS EVENTS
- select Configure actor to edit settings
- Select Pellet_injection ON if you like to use pellet in your simulation
- Select mode of operation: specific - pellets will be shut at specific times, you also need to specify array times_pellet
  - specific - pellets will be shut at exact times specified in array times_pellet
  - frequency pellets will be shut from tstart_pellet until tend_pellet with a frequency_pellet
- frequency pellets will be shut from tstart_pellet until tend_pellet with a frequency_pellet
- Commit



Parameters of individual pellet need to be configured through the icode_parameters of the PELLET actor. To access it go to 'Outline' on the right upper corner and open the following:



- right click on the actor PELLET
- select Configure actor
- click Edit Code Parameters
- edit parameters and click save and exit
- Commit



amn atomic mass number: array of elements separated by space (1:nelements) [-]

zn nuclear charge: array of elements separated by space (1:nelements) [-]

fraction fraction of each element in the pellet, based on the number of atoms: array of elements separated by space (1:nelements) [-]

rpell radius of the pellet [m]

vpell velocity of the pellet [m/s]

rcloud radius of the pellet cloud [m], radial extension of the cloud = 2*rp0

lcloud length of the pellet cloud along the field line [m]

Tcloud temperature of the pellet cloud [eV]

Pellet path is specified by two points, for which R and Z coordinated should be specified

R R coordinates of the pivot and second points of the pellet path, separated by space [m]

Z Z coordinates of the pivot and second points of the pellet path, separated by space [m]

Control switches allow to activate:

- ***drifts*** - YES - will activate radial displacement of deposition profile, same for all path points
- ***cooling*** - YES - will activate cooling of the other side of the plasma due to parallel heat transport (essential for large pellets, which might cross the same flux surface twice)
- ***JINTRAC*** - YES - will provide temperature reduction consistent with the model used in JETTO

2.2.4.9.2 MHD

At the top level of the workflow you can switch ON/OFF possible MHD events

- right click on the box INSTANTANEOUS EVENTS
- select Configure actor to edit settings
- Select SAWTOOTH ON if you like to use them in your simulation
- Commit

2.2.4.10 Visualization during the run

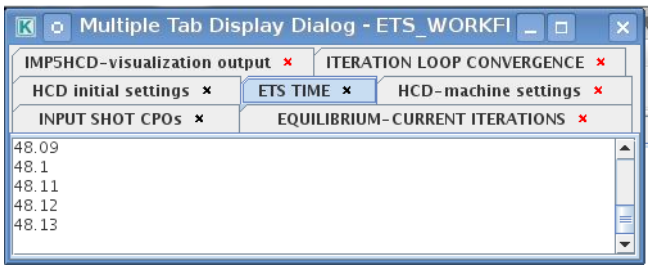
There is a number tools visualizing the ETS run.

2.2.4.10.1 Multiple Tab Display

The display appeares automaticaly when the ETS workflow is launched. It displays diagnostic text messages from the workflow on following topics:

- Input data statement
- Iterations to check the initial convergence between EQUILIBRIUM and CURRENT
- Time evolution
- Convergence of iteratinos within the time step
- IMP5HCD settings
- Power used by IMP5HCD actors durung the run

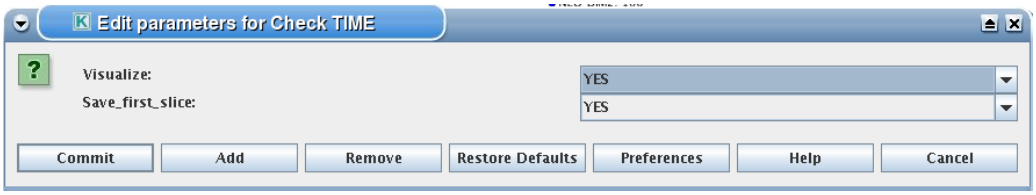
Also the error messages from execution of the workflow will be displayed here.



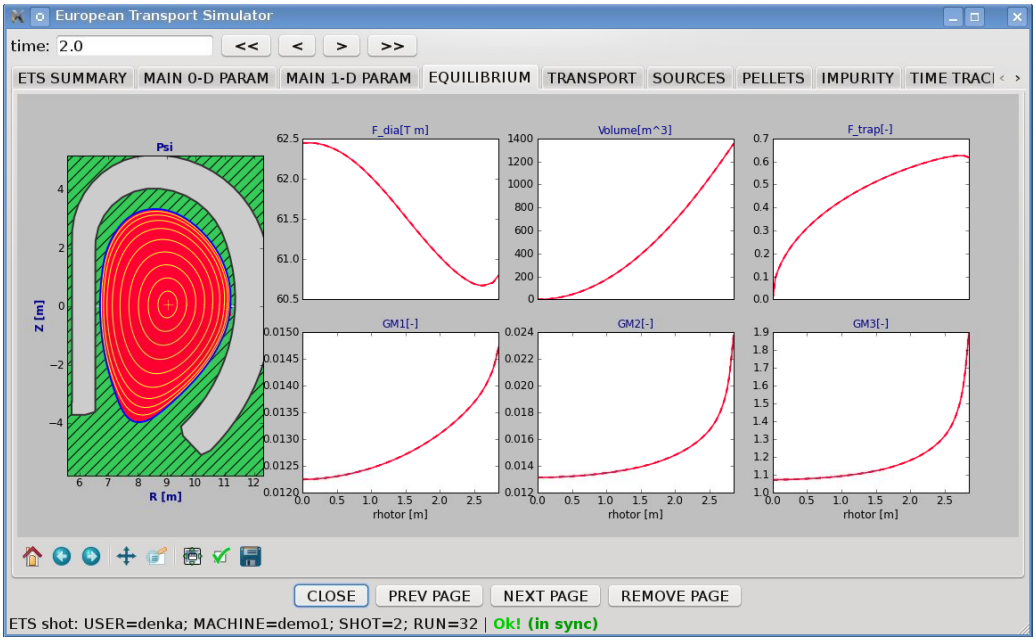
2.2.4.10.2 Python Visualization Display

You can activate the graphical visualization of your run evolution:

- right click on the box Check Time & Save Slice
- select Configure actor
- select visualisation YES or NO
- Commit



Then evolution of main discharge parameters will be shown in this window:



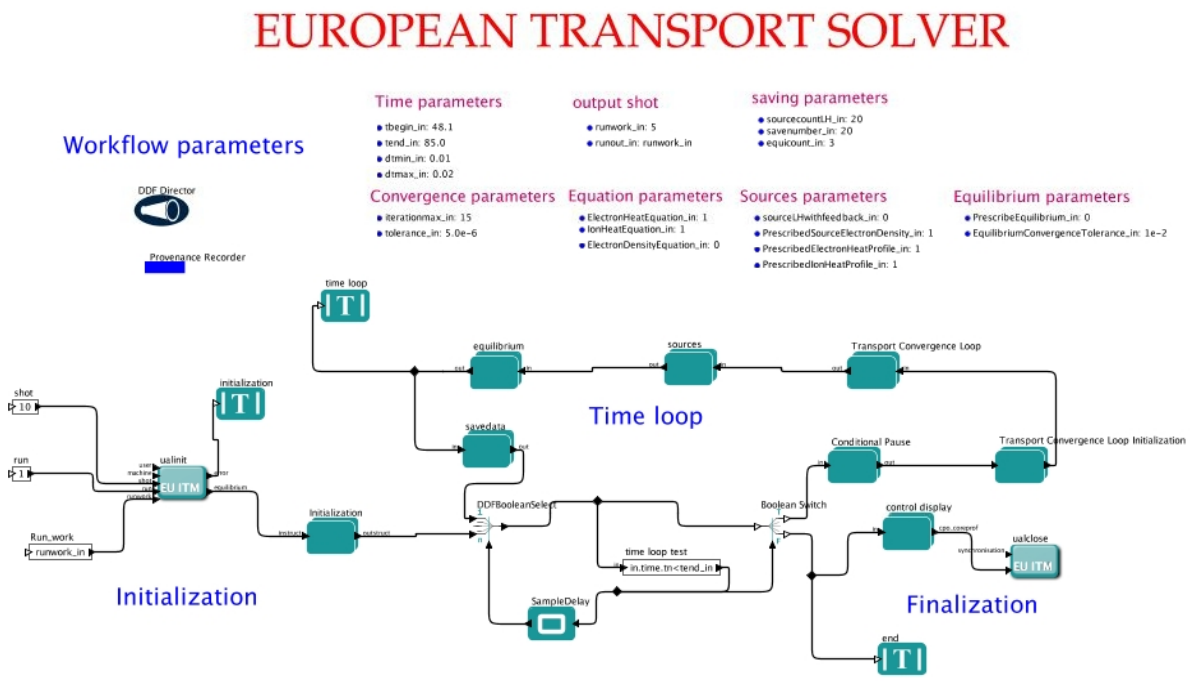
2.3 ETS_C

The ETS workflow (IMP3-ACT1) is used for 1-D transport simulation of a tokamak core plasma.

ETS workflow in KEPLER :

- uses as actors and composite actors from other IMPs, thus for the most recent versions of them please check with relevant project
- complex, but clearly structured workflow, which offers user friendly interface for configuring the simulation
- allows for easy modifications (*connecting new modules, or reconnecting the parts of the workflow*) through the easy graphical interface
- provides users with all updates through the version control system
- still actively developing tool

Contact persons: vincent.basiuk@cea.fr?subject=ETS%20in%20KEPLERphilippe.huynh@cea.fr?subject=ETS%20in%20KEPL



2.3.1 Trainings

Here is the training given in 2011 at GARCHING : Introduction training 2011, [58] training 2011 [59]

2.3.2 Download version of ETS_C workflows and actors

- INNSBRUCK 2011, see the README file in ~huynh/public/INNSBRUCK2011
- GARCHING 2011, see the README file in ~huynh/public/GARCHING2011

---

last update: 2012-07-18 by coster

2.4 ETS Status

Package Name / Physics Module	*ETS-A* [60]	*ETS-C* [61]
EQILIBRIUM		
fixed boundary:		
BDSEQ	Ready for use	
EMEQ	Ready for use	
SPIDER	Ready for use	
SPIDER_IMP12	Ready for use	
CHEASE	Ready for use	validate
HELENA	Ready for use	
HELENA21		work in 4.09a problem when it doesn't find any equilibrium crash

[58] https://www.efda-itm.eu/ITM/imports/imp3/public/introduction_ETS_2011.pdf
[59] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_C_training_2011.pdf

Package Name / Physics Module	*ETS-A* [60]	*ETS-C* [61]
free boundary:		
CEDRES++	In progress/tests are planned for Nov.2014	validate (static mode, TBD evolution mode)
CREATE-NL		
FIXFREE		
EQFAST		work in 4.09a
FREEBIE		validate
MHD		
NTM	Ready for use	validate
SAWTEETH	Implemented/Tested/release date:Nov.2014	
Linear Stability Chain	Stand alone tests/implementation in ETS and release:2015	
TRANSPORT		
analytical & interpretative:		
From DATA BASE (interpretative)	Ready for use	
Edge Transport Barried (analytical)	Ready for use	
anomalous:		
ETAIGB	Ready for use	
BOHM-GYROBOHM	Ready for use	validate, + effect of rotation
GLF23	Implemented/Tested/release date:Nov.2014	to be tested (GLF23 installed in previous gateway not validated)
WEILAND	Implemented/Tested/release date:Nov.2014	
RITM	Implemented/Tested/release date:Nov.2014	
EWDM	Implemented/Tested/release date:Nov.2014	
TGLF	In progress/Some initial tests	
KIAUTO		installed (transport model based on scaling law)
neoclassical:		
NEOS	Ready for use	
NEOWES	Ready for use	
NEOART	Ready for use (probably not suggested as being too oscilatory)	
NCLASS	In progress	validate with composition (to be upgrade with compositions
NCLASS/FORCEBALL		installed (gives the radial electric field)

Package Name / Physics Module	*ETS-A* [60]	*ETS-C* [61]
HEAT,PARTICLE SOURCES & CURRENT DRIVE		
analytical & interpretative:		
From DATA BASE (interpretative)	Ready for use	
Gaussian	Ready for use	
impurity and particles:		
IMPURITY	Ready for use	
NEUTRALS	Ready for use	
PELLET	Ready for use	
ZNEUTRES		installed (simple module of CRONOS for neutral source terms)
ZRECYCLE		edge boundary for electron density
ECRH		
GRAY	Ready for use	Installed
TORAY-FOM		In preparation
TRAVIS	Tested	In preparation
TORBEAM		In preparation
ICRH		
TORIC	In progress	In preparation
ICDEP		Installed
FPSIM		Installed
NBI		
NEMO	Ready for use	Installed
BBNBI	Ready for use	In preparation
NBISIM	Ready for use	Installed
ASCOT	Ready for use	
RISK	Ready for use	In preparation
LH		
nuclear sources		
nuclearsim	Ready for use	Installed
CONTROLS		

Package Name / Physics Module	*ETS-A* [60]	*ETS-C* [61]
NBI power control	Ready for use	
ECRH power control	Ready for use	
ICRH power control	Ready for use	
Pellet frequency control	Ready for use	
COUPLING TO EDGE		
SOLPS	Tested at Fortran level	
DOCUMENTATION and MANUALS		
Physics Description	Description of the ETS [62]	
Numerics Description	Form of the standardize equations [63]	
Manuals	• ETS workflows in KEPLER [64] • ETS source in Fortran [65]	

last update: by

last update: 2015-02-03 by denka

3 Documentation for the ETS

- Current ETS Timeline (PDF) [66](MS Project) [67]
- Description of the ETS [68]
- Form of the standardize equations [69]
- ETS User Guide [70]
- ETS Status [71]
- ETS Doxygen Documentation (PDF) [72](HTML) [73]
- Pellets in ETS [74]

4 Presentations that discuss the ETS

- Presentation at ICNSP-2009 on the ETS [75]

[62] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_TRANSPORT_EQUATIONS.pdf
[63] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/STANDARDISED_EQUATION.pdf
[64] https://www.eufus.eu/documentation/ITM/html/ETS_in_KEPLER.html
[65] https://www.eufus.eu/documentation/ITM/html/ETS.html#ETS_2
[66] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_timeline.pdf
[67] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_timeline.mpp
[68] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_TRANSPORT_EQUATIONS.pdf
[69] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/STANDARDISED_EQUATION.pdf
[70] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_User_Guide.pdf
[71] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_Status.pdf
[72] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Documentation/ETS_Doxygen.pdf
[73] https://portal.eufus.eu/documentation/ITM/doxygen/imp3/ets/
[74] https://www.efda-itm.eu/ITM/html/pellet.html
[75] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Presentations/2009_ICNSP/ETS_Coster_ICNSP-2009_v5.ppt

- Movie from the presentation showing the evolution of the flux surfaces [76]
- Movie from the presentation showing the evolution of the plasma [77]

5 ETS Verification & Validation [78]

6 Other ETS related information

- Visualization of the repository activity (x264) [79]
- Visualization of the repository activity (wmv2) [80]

---

last update: 2019-01-31 by g2dpc

[76] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Presentations/2009_ICNSP/psi_5_42.mpg
[77] https://www.efda-itm.eu/ITM/imports/imp3/public/ETS_Presentations/2009_ICNSP/comb_psi_5_42.900x400.mpg
[78] https://www.efda-itm.eu/ITM/html/imp3_ets_vv.html
[79] https://www.efda-itm.eu/ITM/imports/imp3/public/ets.mp4
[80] https://www.efda-itm.eu/ITM/imports/imp3/public/ets.wmv