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 person: Denis Kalupin (Skype: dkalupin)

European Transport Solver

Workflow parameters

DDF Director

General parameters:

- USER: kalupin
- machine: test

- shot_in: 77922
- run_in: 1

- shot_out: 77922
- run_out: 10

Times:

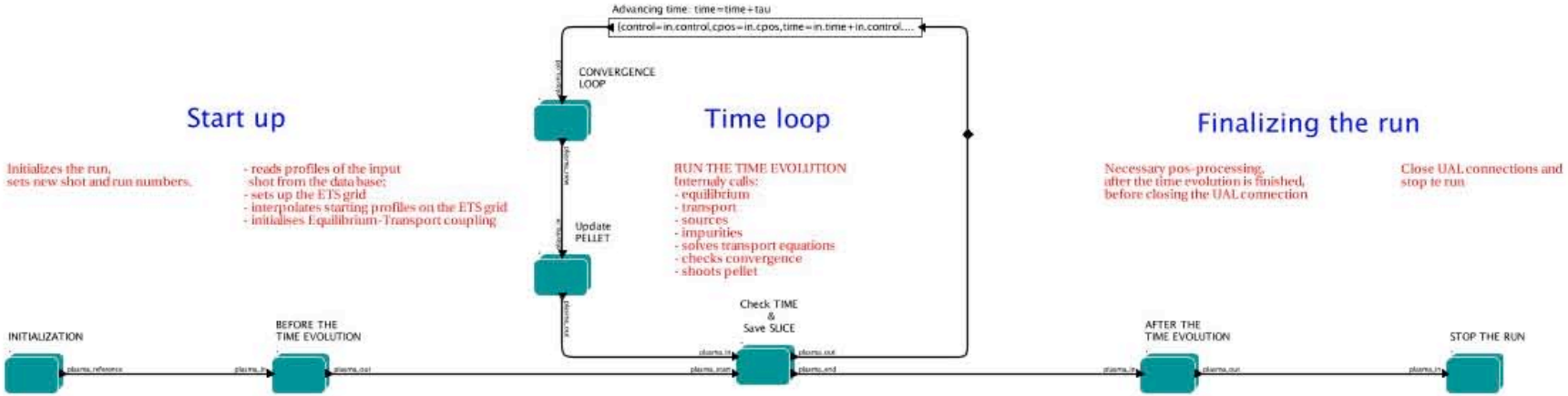
- tbegin_in: 47.7
- tend_in: 47.8

General parameters:

- rho_bnd: 1.23
- r0: 2.96
- b0: 2.36
- ip: 1.7e6

ETS dimensions:

- nrho: 100
- nion: 1
- nimp: 0
- max_nzimp: 1
- nneut: 0
- max_ncomp: 1
- max_mype: 2



Start up

Initializes the run,
sets new shot and run numbers.

- reads profiles of the input shot from the data base;
- sets up the ETS grid
- interpolates starting profiles on the ETS grid
- initialises Equilibrium-Transport coupling

Time loop

RUN THE TIME EVOLUTION
Internaly calls:
- equilibrium
- transport
- sources
- impurities
- solves transport equations
- checks convergence
- shoots pellet

Finalizing the run

Necessary pos-processing,
after the time evolution is finished,
before closing the UAL connection

Close UAL connections and stop te run

INSTRUCTIONS on obtaining the ETS workflow

1. Your environment

If you like to run the latest version of the code, there are certain requirements on the environment, which should be set before executing the code.

a. Make sure that you use data base version 4.09a:

```
>cd
>ls public/itmdb/itm_trees/test/
```

if 4.09a is not present ==> create it:

```
>/afs/efda-itm.eu/project/switm/scripts/create_user_itm_dir test 4.09a
```

b. If your private ITM Database is currently in AFS, it is suggested moving it to the parallel file system.

To check where it is, please execute the following command

```
>df -h ~/public/itmdb/
```

and if you see the following

```
Filesystem            Size  Used Avail Use% Mounted on
AFS                   8.6G     0  8.6G   0% /afs
```

then your database is in AFS and it should be moved. If you see

```
Filesystem            Size  Used Avail Use% Mounted on
ib-pfs1@o2ib1:/efda1   72T   16T   53T  24% /efda1
```

then your database is already on the parallel file system and you should ignore it.

To transfer your private database "itmdb" from AFS to PFS:

```
>cd ~/public
>mkdir -p /pfs/work/$GROUP/$USER
>mv -i itmdb /pfs/work/$GROUP/$USER/itmdb
>ln -s /pfs/work/$GROUP/$USER/itmdb itmdb
```

c. Do source to this data base:

```
>source /afs/efda-itm.eu/project/switm/scripts/ITMv1 kepler test 4.09a
```

2. Obtaining the ETS

a. Copy input JET shot 77922 into your database:

>svn co http://gforge.efda-itm.eu/svn/keplerworkflows/trunk/4.09a/imp3/ets/DATA $MDSPLUS_TREE_BASE_0

b. Copy the ETS actors and workflow to your space:

>svn co http://gforge.efda-itm.eu/svn/kepleractors/trunk/4.09a/imp3/ets ETS_ACTORS

>svn co http://gforge.efda-itm.eu/svn/keplerworkflows/trunk/4.09a/imp3/ets ETS_WORKFLOWS

>svn co http://gforge.efda-itm.eu/svn/keplerworkflows/trunk/4.09a/imp3/ets/kplots/ ~/kepler/kplots/

c. Compile ETS actors:

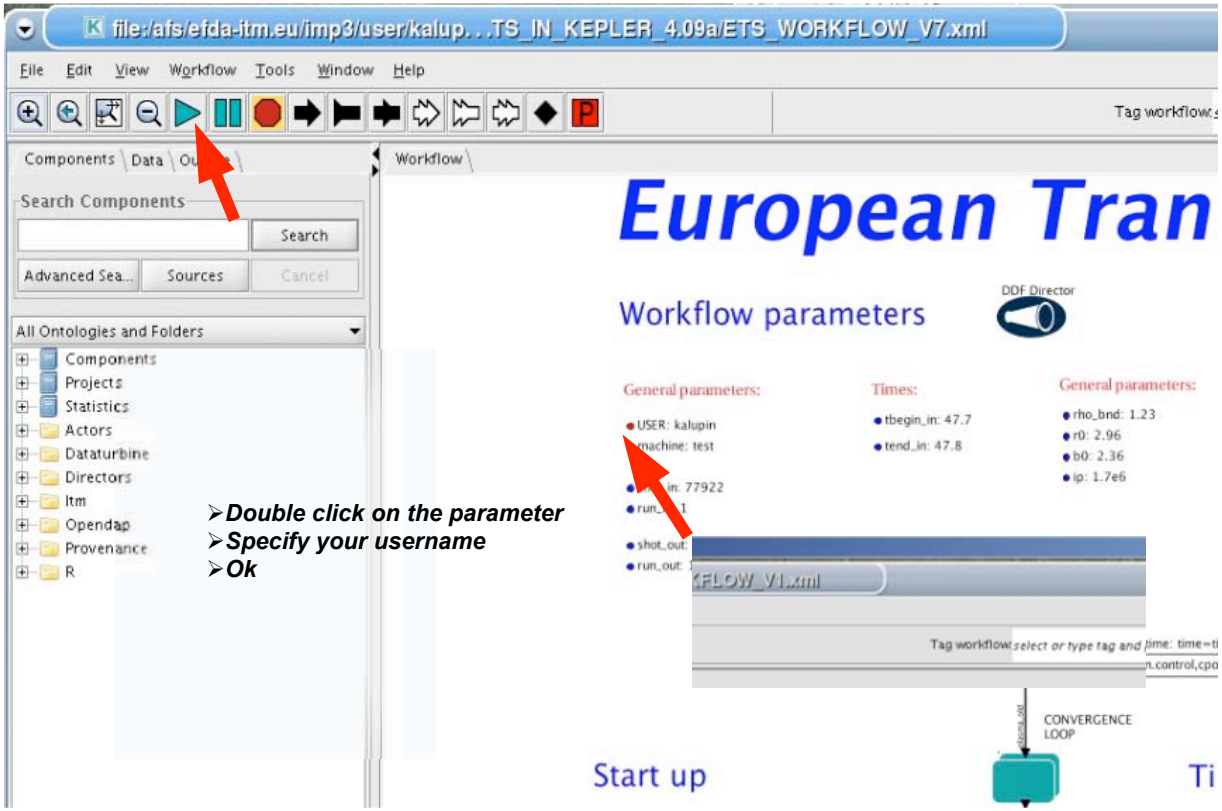
```
>cd ETS_ACTORS
>import_ets_actors
```

d. Compile actors from other IMPs:

!!!! Preferably, official versions have to be used,
thus, check it from other IMPs out. As fast solution use the following script,
but it is not necessary the most recent version of actors.

>import_imps_actors

3. Executing the ETS



a. open ETS workflow in Kepler:

>kepler.sh ~/ETS_WORKFLOWS/ETS_WORKFLOW_V3.xml

- on the top of the workflow, change the parameter "user" to your user_ID.

b. you can run the workflow!!!

Special notes:

- If you like to run impurity simulations, swithch to the input shot 71827 run 1, change shot parameters and update your AMNS data (instructions on: amns_data_providers_documentation)

- There is a version of the workflow including HCD actors available. To run it, following steps are necessary:

- copy hcd actors to your KEPLER (detailed instructions on: IMP5-HCD)

```
>cd
>svn co http://gforge.efda-itm.eu/svn/keplerworkflows/trunk/4.09a/imp5/imp5hcd IMP5HCD
>cd IMP5HCD
>import_hcd_actors
```

-Open ETS workflow with HCD:

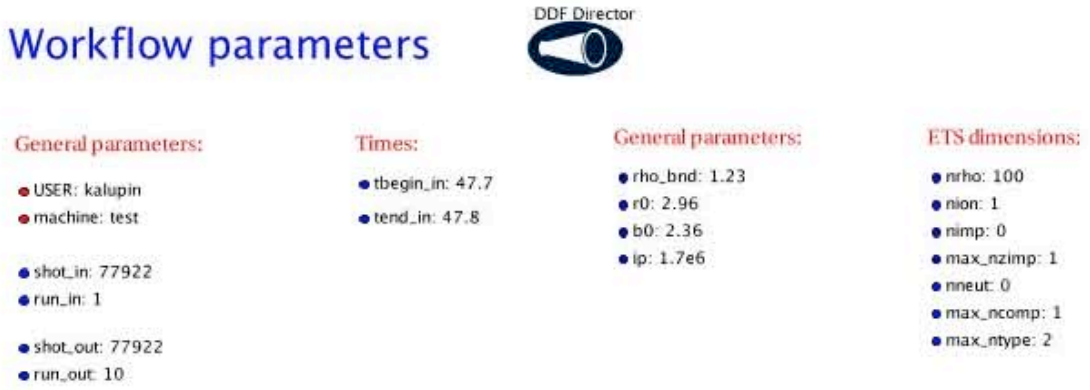
```
>kepler.sh ~/ETS_WORKFLOWS/ETS_WORKFLOW_V2.xml
```

4. Configuring your run

You have a number of options to configure your run:

- workflow parameters

There is a number of parameters used at different levels of the workflow, each of them is visible from the specified level down. You can change the parameter value by double click on it.

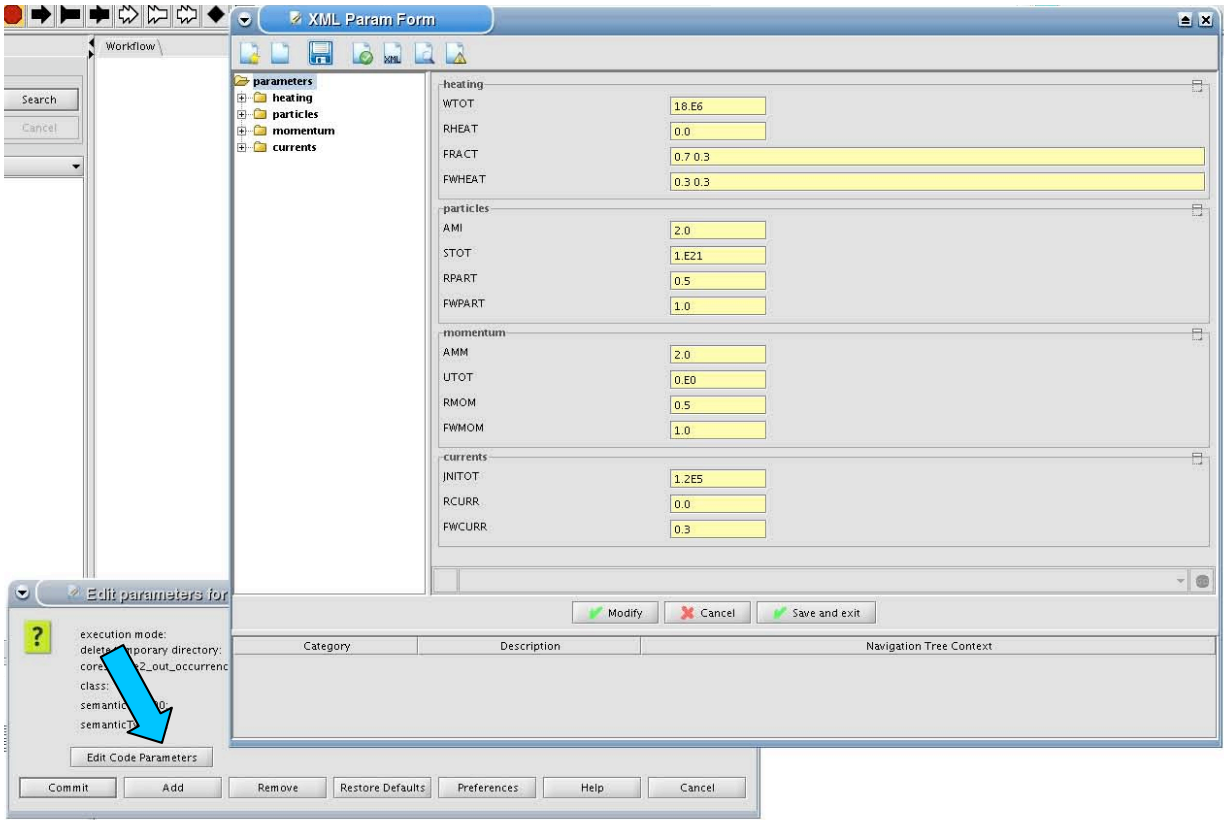


- control switches

Most of composite actors contain a number of switches. You can access them by right click on the actor, select 'Configure actor'. Do necessary changes (select required modules, change the numbers, etc.) and click 'Commit' to save your changes.

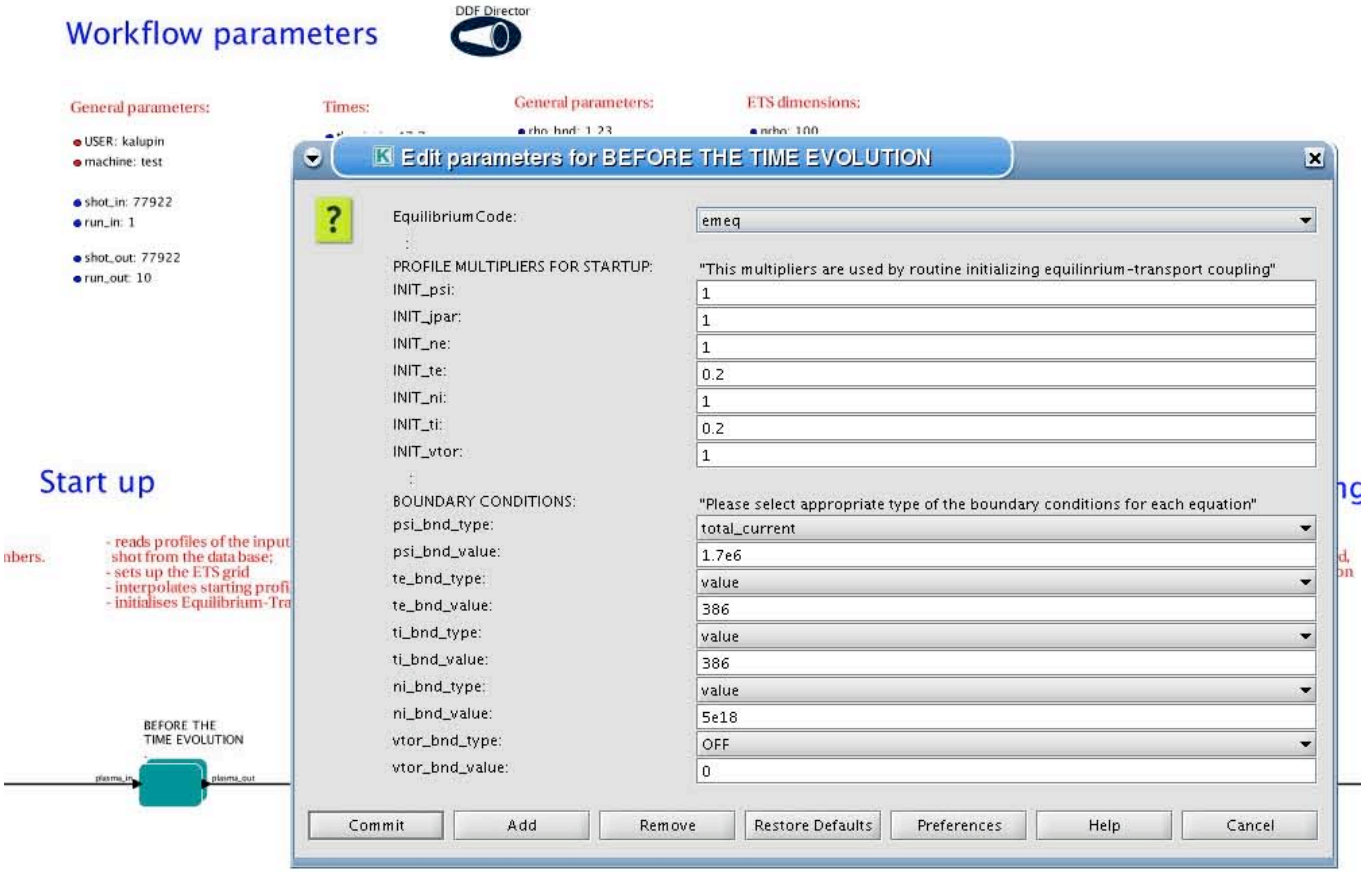
- XML parameters attached to the actor

    Some of actors have individual settings, specified through the XML file. You can change them by right click on the actor, select ‘Configure actor’, select ‘Edit Code Parameters’, do necessary changes, click ‘Save and exit’, click ‘Commit’


XML Param Form
Workflow
Search
Cancel
parameters
heating
particles
momentum
currents
heating
WTOT
18.E6
RHEAT
0.0
FRACT
0.7 0.3
FWHEAT
0.3 0.3
particles
AMI
2.0
STOT
1.E21
RPART
0.5
FWPART
1.0
momentum
AMM
2.0
UTOT
0.E0
RMOM
0.5
FWMOM
1.0
currents
JNITOT
1.2E5
RCURR
0.0
FWCURR
0.3
Modify
Cancel
Save and exit
Category
Description
Navigation Tree Context
Edit parameters for
execution mode:
class:
Edit Code Parameters
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel

EQUATIONS TO BE SOLVED AND BOUNDARY CONDITIONS:

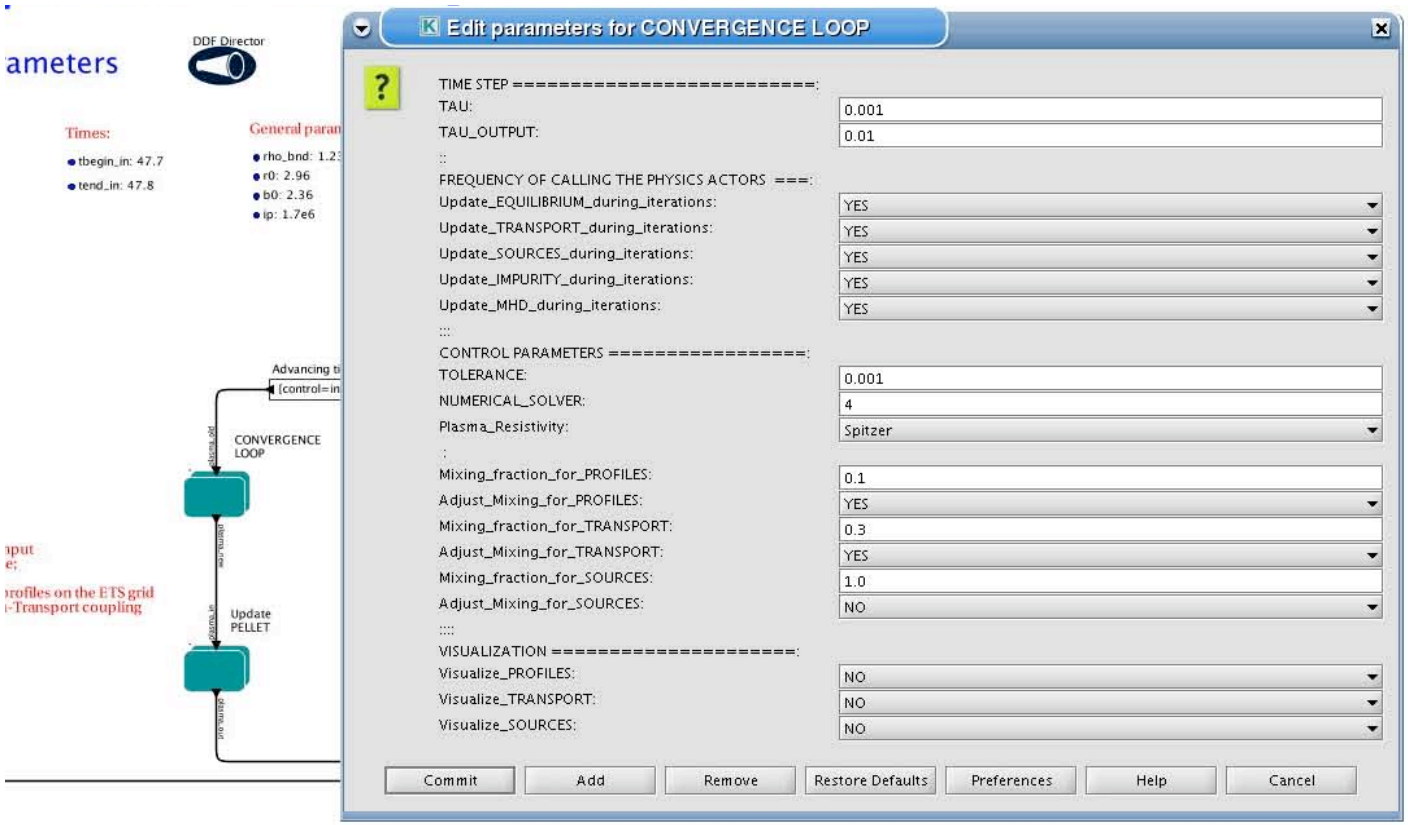
- *right click on the box 'BEFORE THE TIME EVOLUTION'*
- *select 'Configure actor'*
- *select appropriate boundary condition for each equation (OFF-equation is not solved)*
- *specify values for boundary conditions corresponding to the type*
- *Commit*


Workflow parameters
DDF Director
General parameters:
USER: kalupin
machine: test
shot_in: 77922
run_in: 1
shot_out: 77922
run_out: 10
Times:
General parameters:
rho_bnd: 1.23
ETS dimensions:
nrho: 100
Start up
- reads profiles of the input
shot from the data base;
- sets up the ETS grid
- interpolates starting profi
- initialises Equilibrium-Tra
BEFORE THE
TIME EVOLUTION
plasma_in
plasma_out
Edit parameters for BEFORE THE TIME EVOLUTION
EquilibriumCode:
emeq
PROFILE MULTIPLIERS FOR STARTUP:
"This multipliers are used by routine initializing equilinrium-transport coupling"
INIT_psi:
1
INIT_jpar:
1
INIT_ne:
1
INIT_te:
0.2
INIT_ni:
1
INIT_ti:
0.2
INIT_vtor:
1
BOUNDARY CONDITIONS:
"Please select appropriate type of the boundary conditions for each equation"
psi_bnd_type:
total_current
psi_bnd_value:
1.7e6
te_bnd_type:
value
te_bnd_value:
386
ti_bnd_type:
value
ti_bnd_value:
386
ni_bnd_type:
value
ni_bnd_value:
5e18
vtor_bnd_type:
OFF
vtor_bnd_value:
0
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel

GENERAL SETTINGS FOR THE RUN:

- *right click on the box 'CONVERGENCE LOOP'*
- *select 'Configure actor' to edit settings*
- *Select your settings or specify values*
- *Commit*

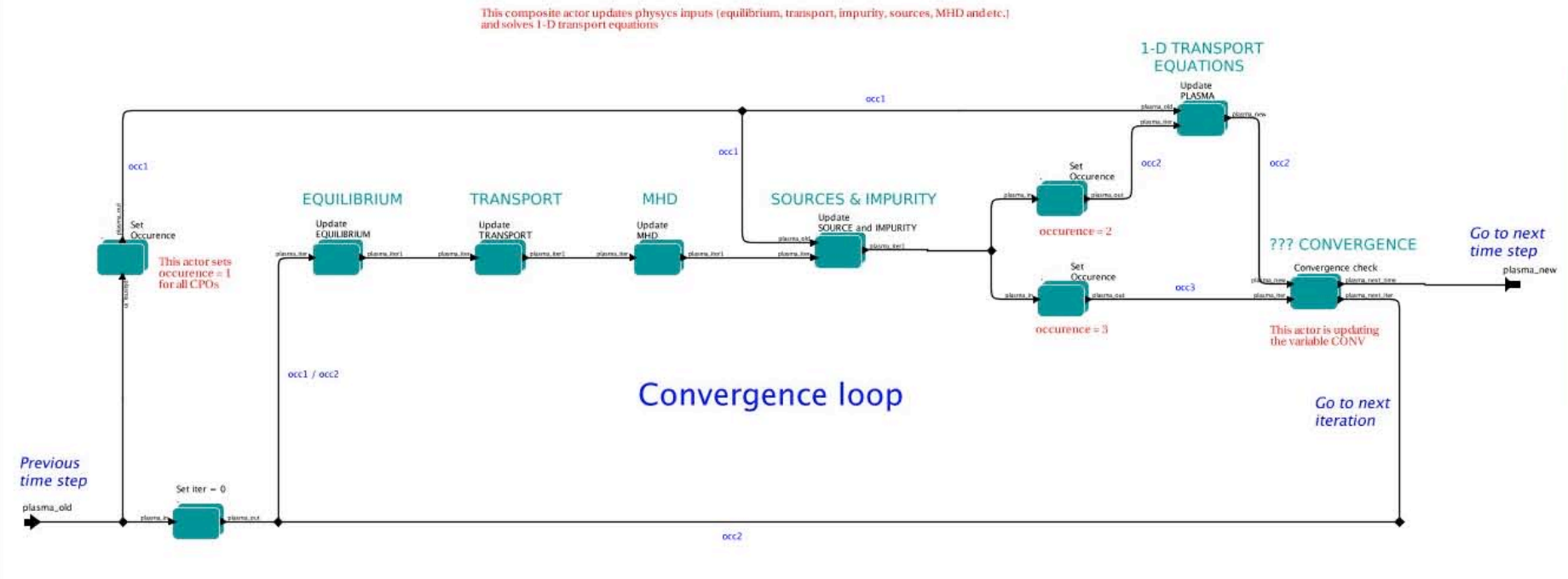
TAU - *time step*
TAU_OUTPUT - *time step for output to the data base*
FREQUENCY OF CALLING THE PHYSICS ACTORS – *for any of them 'YES' means that the relevant actor will be called every iteration; 'NO' means that the relevant actor will be called once in the time step (the first iteration only)*
TOLERANCE - *precision you need to follow in your simulations*
NUMERICAL_SOLVER - *choice of numerical scheme used to solve transport equations (do not change without consulting to the ETS team)*
PLASMA_RESISTIVITY - *choice of model for plasma resistivity*
Mixing_fraction_... – *fraction of new solution for relevant quantity adopted during iterations: n(i)=n(i)*F+n(i-1)*(1-F)*
Adjust_Mixing_... – *if 'YES' the ETS will adjust the the Mixing_fraction during iterations for faster convergence; If 'NO' the value specified above will be kept constant*
VISUALISATION – *if 'YES' the ETS will plot profiles of relevant quantities on the screen during iterations*



CONVERGENCE LOOP:

- ***right click on the box 'CONVERGENCE LOOP'***
- ***select 'Open actor'***

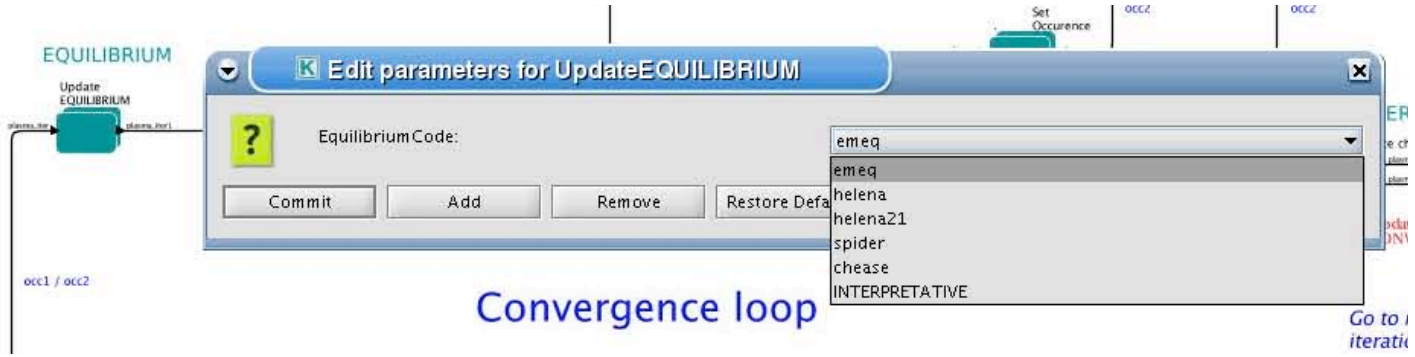
You are inside the convergence loop.
Here you can manage the physics actors attached to your run.



EQUILIBRIUM:

- ***right click on the box 'EQUILIBRIUM'***
- ***select 'Configure actor' to edit settings***
- ***choose one of available equilibrium solver***

- ***Commit***



There are several equilibrium solvers connected to the ETS. You can select the one of them. 'INTERPRETATIVE' means that the ETS will not update the equilibrium, instead it will be using the equilibrium from the input shot/run.
Please also 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.

TRANSPORT:

- ***right click on the box 'TRANSPORT'***
- ***select 'Configure actor' to edit settings***

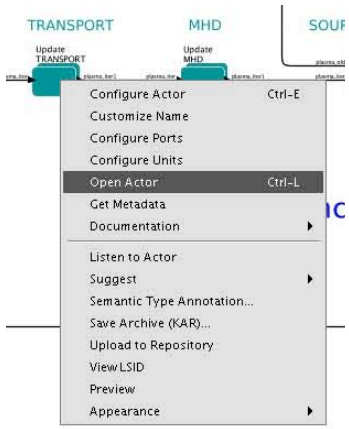
- ***choose your settings***
- ***Commit***

***Bacground Transport** – 'ON' means that ETS will pick up the profiles of transport coefficients saved to your input shot/run; 'OFF'-ignored*
***Anomalous Transport** – You can select one from modules described in the interface (for the particular settings of selected model, go to relevant actor and edit XML parameters); 'OFF'-ignored*
***Neoclassical Transport** – You can select one from modules described in the interface (for the particular settings of selected model, go to relevant actor and edit XML parameters); 'OFF'-ignored*
***Multiplier number for any transport model** – profiles of all transport coefficients from relevant model will be multiplied with specified value*

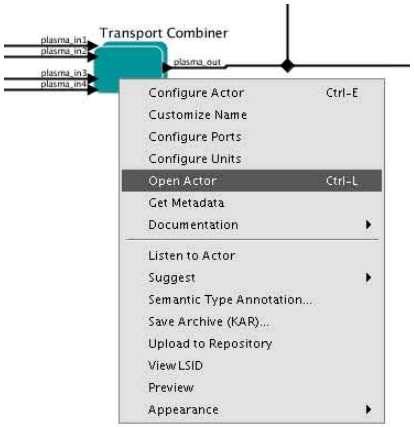
Resulting transport coefficients will be obtained with the formula: D_tot = D_BT*BTM + D_AT*ATM + D_NT*NTM

For more detailed mixture of transport coefficients, please edit XML settings of transport combiner actor:

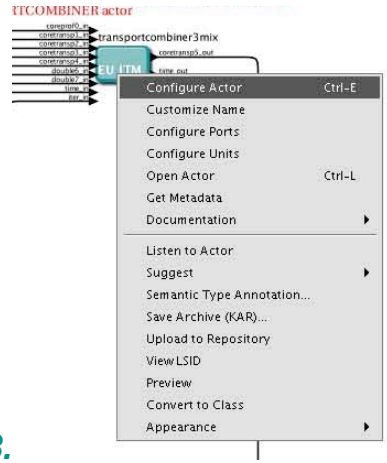
- ***right click on the box 'TRANSPORT'***
- ***select 'Open actor' to edit settings***
- ***right click on the box 'Transport Combiner'***
- ***select 'Open actor' to edit settings***
- ***right click on the box 'transportcombiner3mix'***
- ***select 'Configure actor'***
- ***click 'Edit Code Parameters'***
- ***change coefficients for combining contributions from different transport models (first position- background transport, second position – anomalous transport, third position – neoclassical transport)***
- ***save and exit***
- ***Commit***



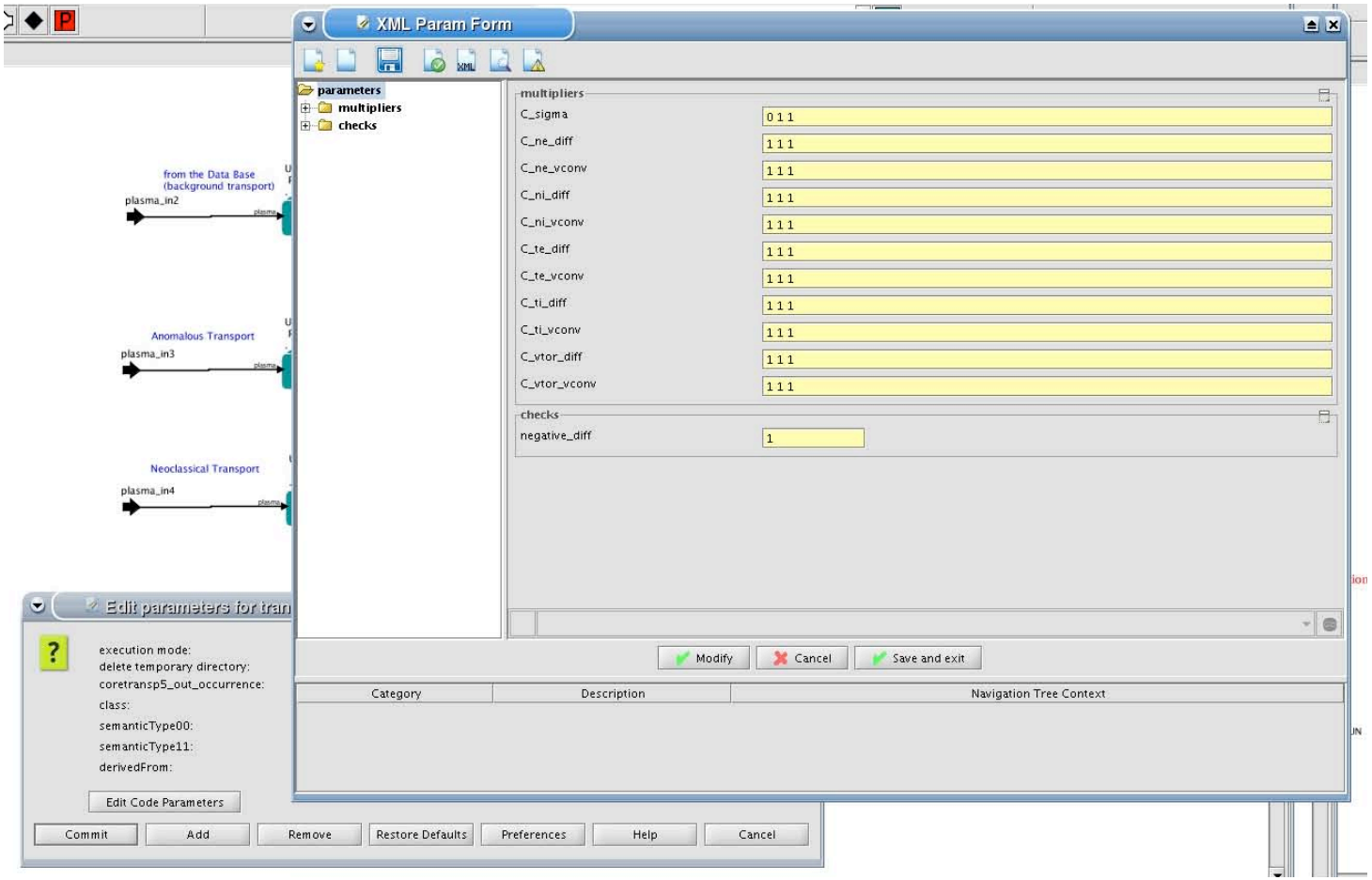
1.



2.



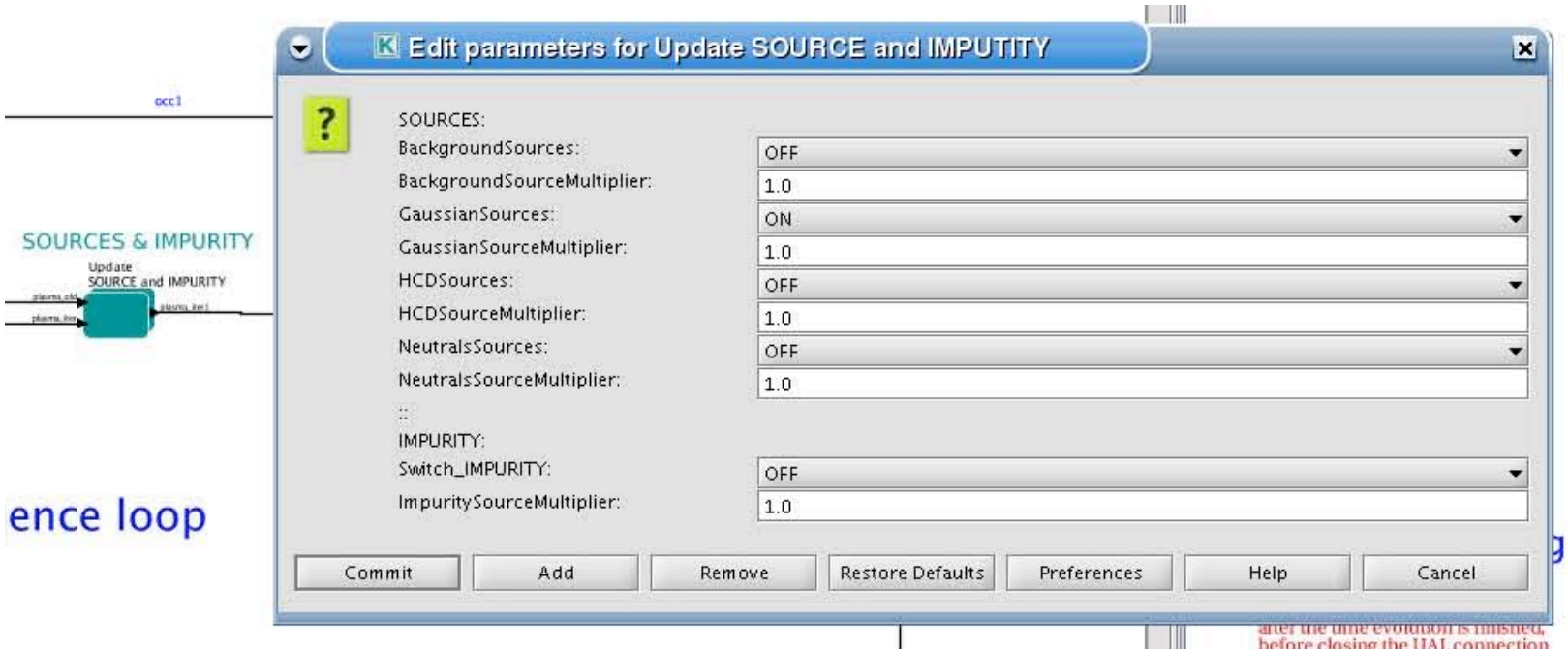
3.


XML Param Form
parameters
multipliers
checks
multipliers
C_sigma
0 1 1
C_ne_diff
1 1 1
C_ne_vconv
1 1 1
C_ni_diff
1 1 1
C_ni_vconv
1 1 1
C_te_diff
1 1 1
C_te_vconv
1 1 1
C_ti_diff
1 1 1
C_ti_vconv
1 1 1
C_vtor_diff
1 1 1
C_vtor_vconv
1 1 1
checks
negative_diff
1
Modify
Cancel
Save and exit
Category
Description
Navigation Tree Context
from the Data Base
(background transport)
plasma_in2
Anomalous Transport
plasma_in3
Neoclassical Transport
plasma_in4
Edit parameters for tran
execution mode:
delete temporary directory:
coretransp5_out_occurrence:
class:
semanticType00:
semanticType11:
derivedFrom:
Edit Code Parameters
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel

4.

SOURCES AND IMPURITY:

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



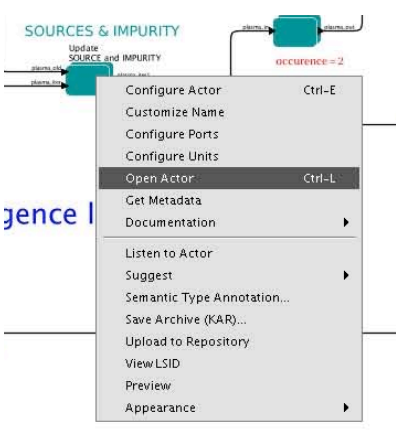
***Bacground Sources** – 'ON' means that ETS will pick up the profiles of sources saved to your input shot/run; 'OFF'-ignored*
***Gaussian Sources** – 'ON' means that ETS will pick up profiles of sources from the Gaussian source actor; 'OFF'-ignored*
***HCD Sources**– 'ON' means that ETS will pick up profiles of sources from the IMP5 actors; 'OFF'-ignored*
***Neutral Sources**– 'ON' means that ETS will pick up profiles of sources from the fluid neutrals actor; 'OFF'-ignored*

Switch_IMPURITY *– 'ON' means that ETS will treat selected impurities; 'OFF'-ignored; 'INTERPRETATIVE' – profiles of impurity density will be read from input shot/run*
Multiplier number for any source *– profiles of all sources from relevant model will be multiplied with specified value*

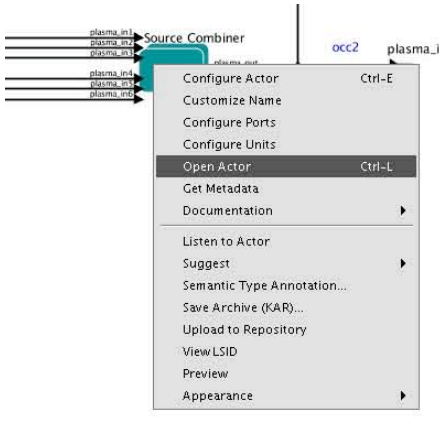
Resulting profiles of sources will be obtained with the formula: S_tot = S_BS*BSM + S_GS*GSM + S_HCD*HCDSM+ S_Neu*NeuSM+ S_IMP*IMPSM

For more detailed mixture of sources, please edit XML settings of source combiner actor:

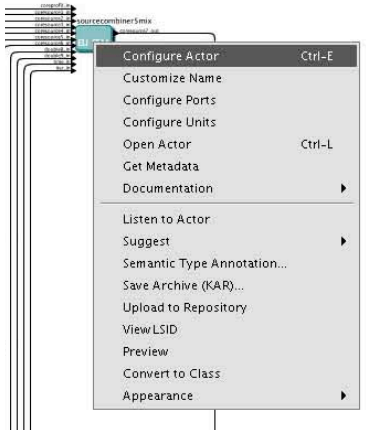
- ***right click on the box 'SOURCES & IMPURITY'***
- ***select 'Open actor' to edit settings***
- ***right click on the box 'Source Combiner'***
- ***select 'Open actor' to edit settings***
- ***right click on the box 'sourcecombiner5mix'***
- ***select 'Configure actor'***
- ***click 'Edit Code Parameters'***
- ***change coefficients for combining contributions from different transport models (first position- background sources, second position – Gaussian sources, third position – HCD sources; fourth position – neutral sources; fifth position – impurity sources)***
- ***save and exit***
- ***Commit***


SOURCES & IMPURITY
Update
SOURCE and IMPURITY
occurence = 2
Configure Actor Ctrl-E
Customize Name
Configure Ports
Configure Units
Open Actor Ctrl-L
Get Metadata
Documentation
Listen to Actor
Suggest
Semantic Type Annotation...
Save Archive (KAR)...
Upload to Repository
View LSID
Preview
Appearance

1.

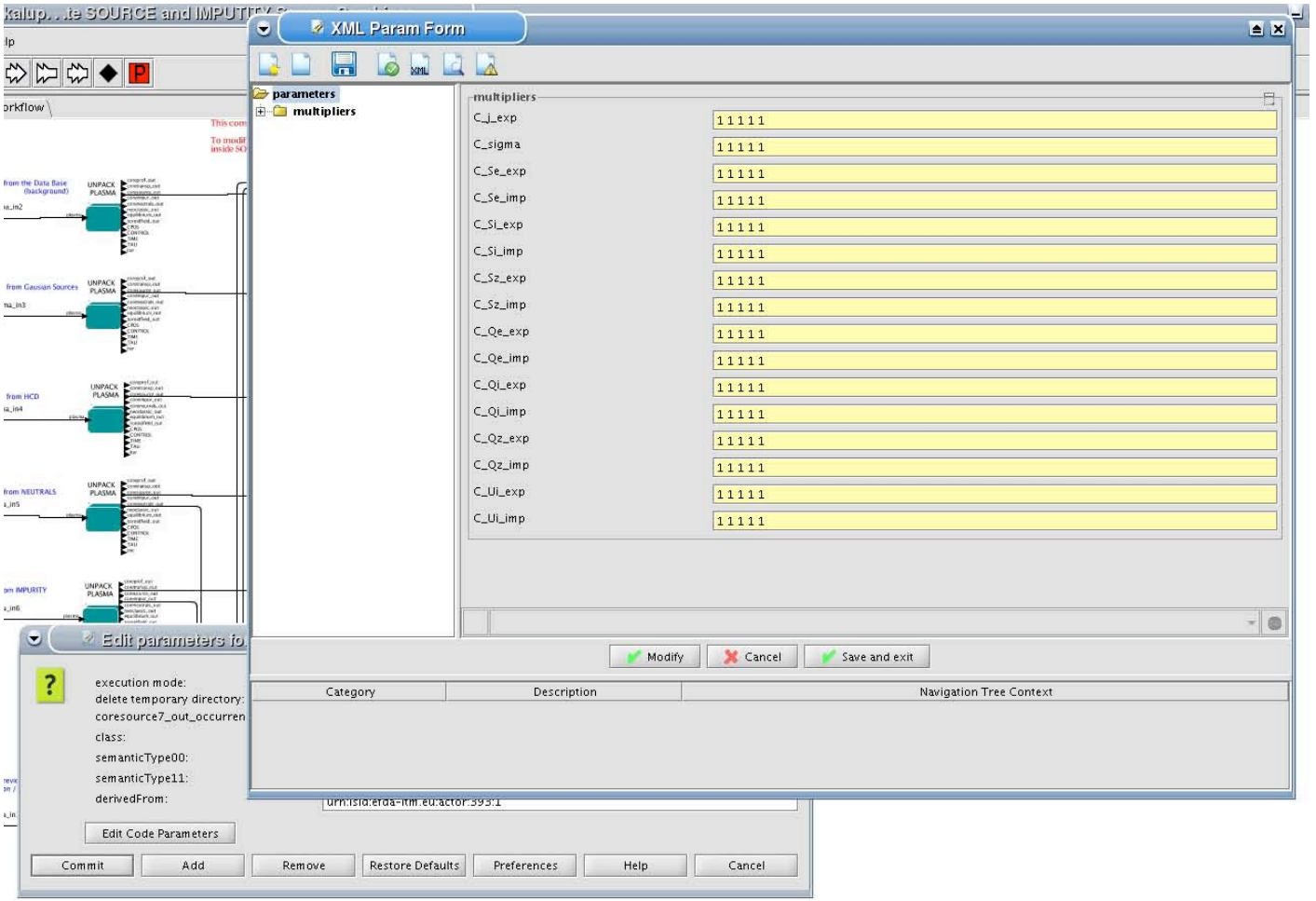

Source Combiner
occ2
plasma_i
Configure Actor Ctrl-E
Customize Name
Configure Ports
Configure Units
Open Actor Ctrl-L
Get Metadata
Documentation
Listen to Actor
Suggest
Semantic Type Annotation...
Save Archive (KAR)...
Upload to Repository
View LSID
Preview
Appearance

2.


Configure Actor Ctrl-E
Customize Name
Configure Ports
Configure Units
Open Actor Ctrl-L
Get Metadata
Documentation
Listen to Actor
Suggest
Semantic Type Annotation...
Save Archive (KAR)...
Upload to Repository
View LSID
Preview
Convert to Class
Appearance

3.

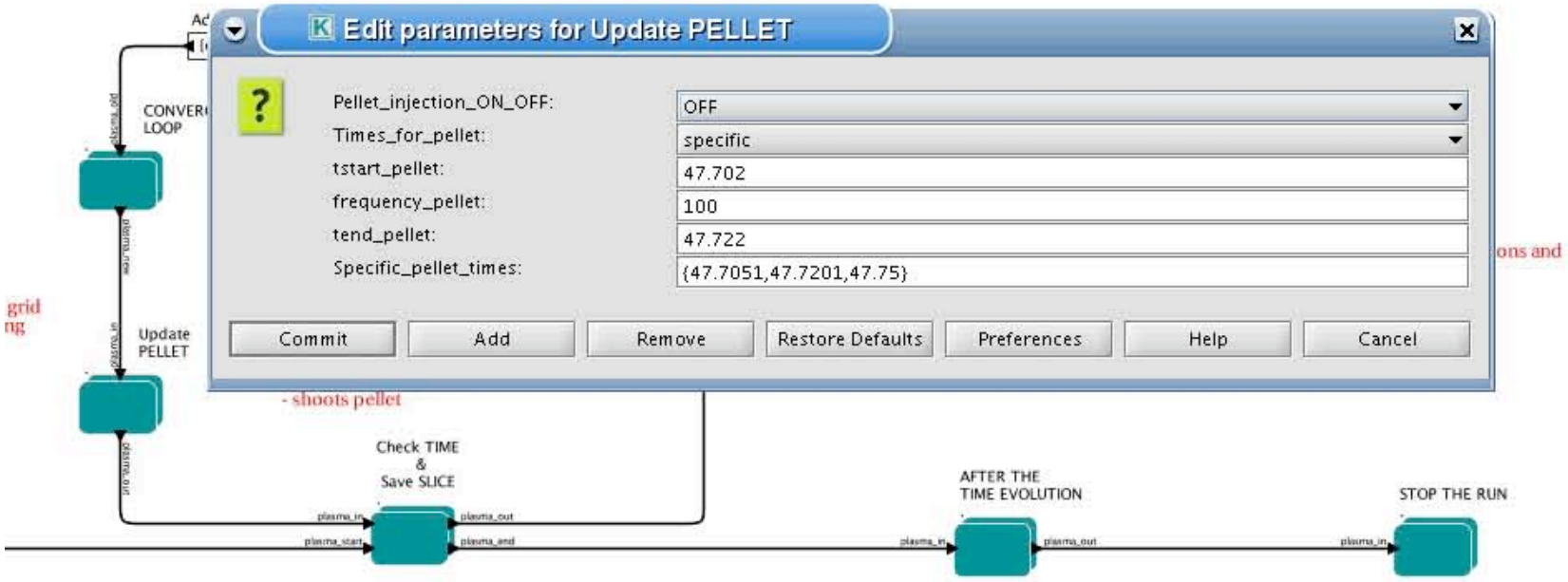
4.


XML Param Form
parameters
multipliers
C_j_exp 1 1 1 1 1
C_sigma 1 1 1 1 1
C_Se_exp 1 1 1 1 1
C_Se_imp 1 1 1 1 1
C_Si_exp 1 1 1 1 1
C_Si_imp 1 1 1 1 1
C_Sz_exp 1 1 1 1 1
C_Sz_imp 1 1 1 1 1
C_Qe_exp 1 1 1 1 1
C_Qe_imp 1 1 1 1 1
C_Qi_exp 1 1 1 1 1
C_Qi_imp 1 1 1 1 1
C_Qz_exp 1 1 1 1 1
C_Qz_imp 1 1 1 1 1
C_Ui_exp 1 1 1 1 1
C_Ui_imp 1 1 1 1 1
Modify
Cancel
Save and exit
Category
Description
Navigation Tree Context
Edit parameters fo
execution mode:
delete temporary directory:
coresource7_out_occurren
class:
semanticType00:
semanticType11:
derivedFrom:
Edit Code Parameters
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel

PELLET:

At the top level of the workflow you can configure Pellet module

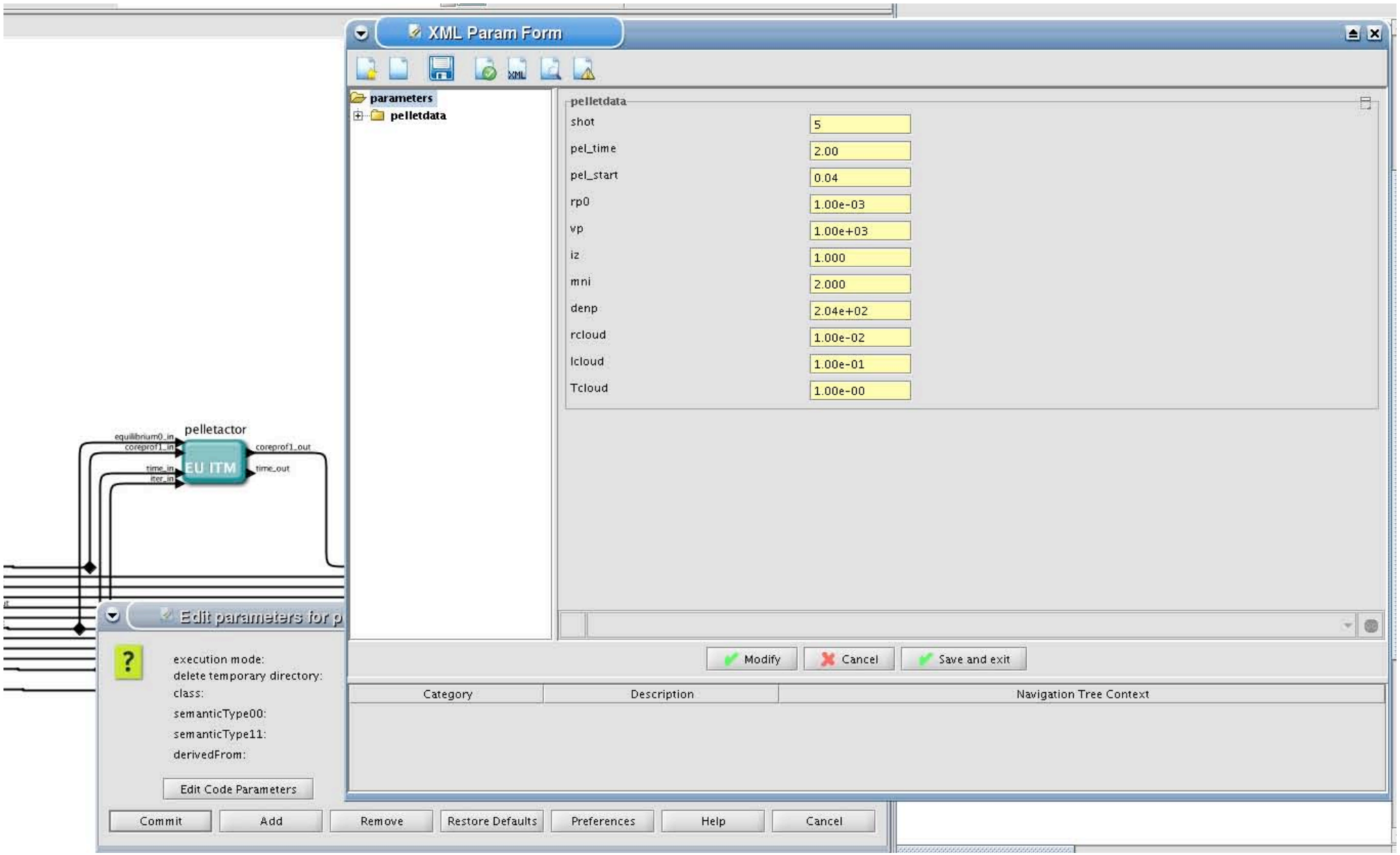
- ***right click on the box 'Update PELLET'***
- ***select 'Configure actor' to edit settings***
- ***Select 'ON' if you like to use pellet in your simulation***
- ***Select mode of operation: 'specific' - pellets will be shut at specific times, you also need to specify array 'times_pellet'***
  'frequency'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



You also can edit parameters of the pellet:

- ***right click on the box 'Update PELLET'***
- ***select 'Open actor'***
- ***right click on the box 'Pellet injection'***
- ***select 'Open actor'***
- ***right click on the box 'pelletactor'***
- ***select 'Configure actor'***
- ***click 'Edit Code Parameters'***

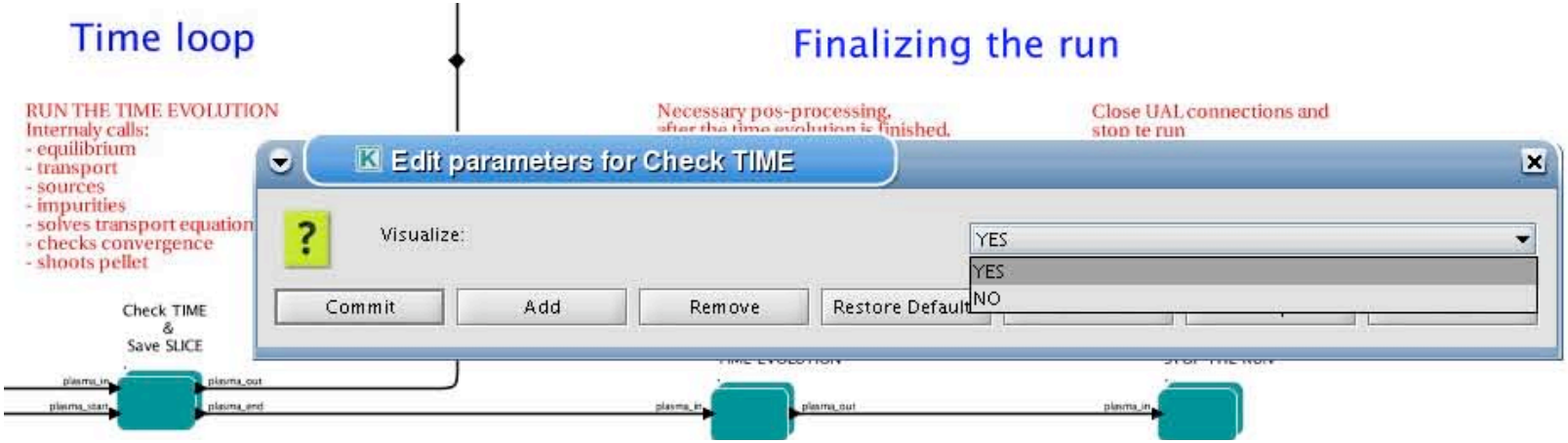
- ***edit parameters and click ‘save and exit’***
- ***Commit***


XML Param Form
parameters
pelletdata
pelletdata
shot 5
pel_time 2.00
pel_start 0.04
rp0 1.00e-03
vp 1.00e+03
iz 1.000
mni 2.000
denp 2.04e+02
rcloud 1.00e-02
lcloud 1.00e-01
Tcloud 1.00e-00
Modify
Cancel
Save and exit
Category
Description
Navigation Tree Context
pelletactor
equilibrium0_in
coreprof1_in
time_in
iter_in
EU ITM
coreprof1_out
time_out
Edit parameters for p
execution mode:
delete temporary directory:
class:
semanticType00:
semanticType11:
derivedFrom:
Edit Code Parameters
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel

Visualization during the run

There is a number of PYTHON visualisation scripts inserted in the workflow. You can activate/deactivate them in several places.

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



This script will visualise a status of the run at the end of each time step:

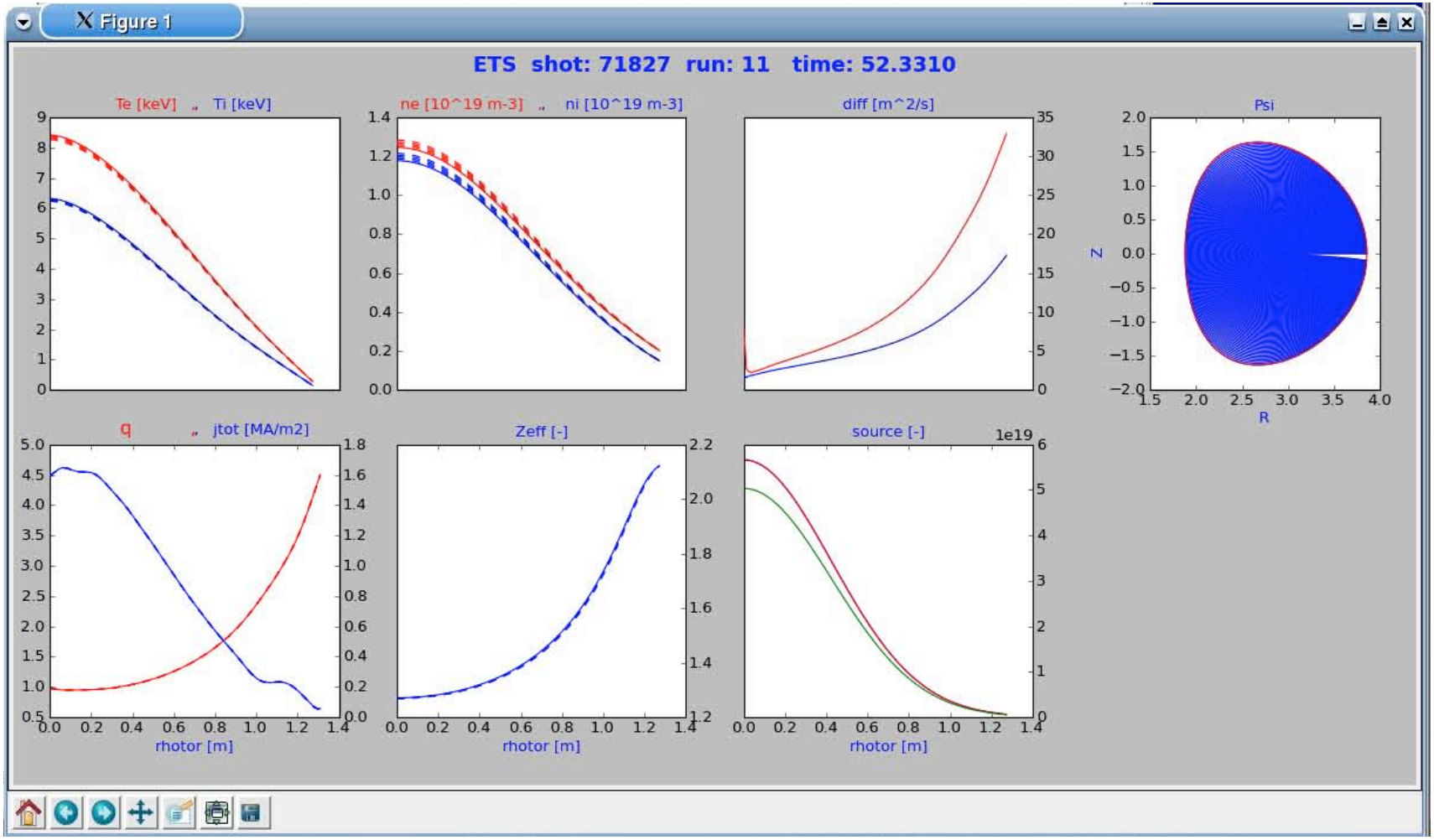
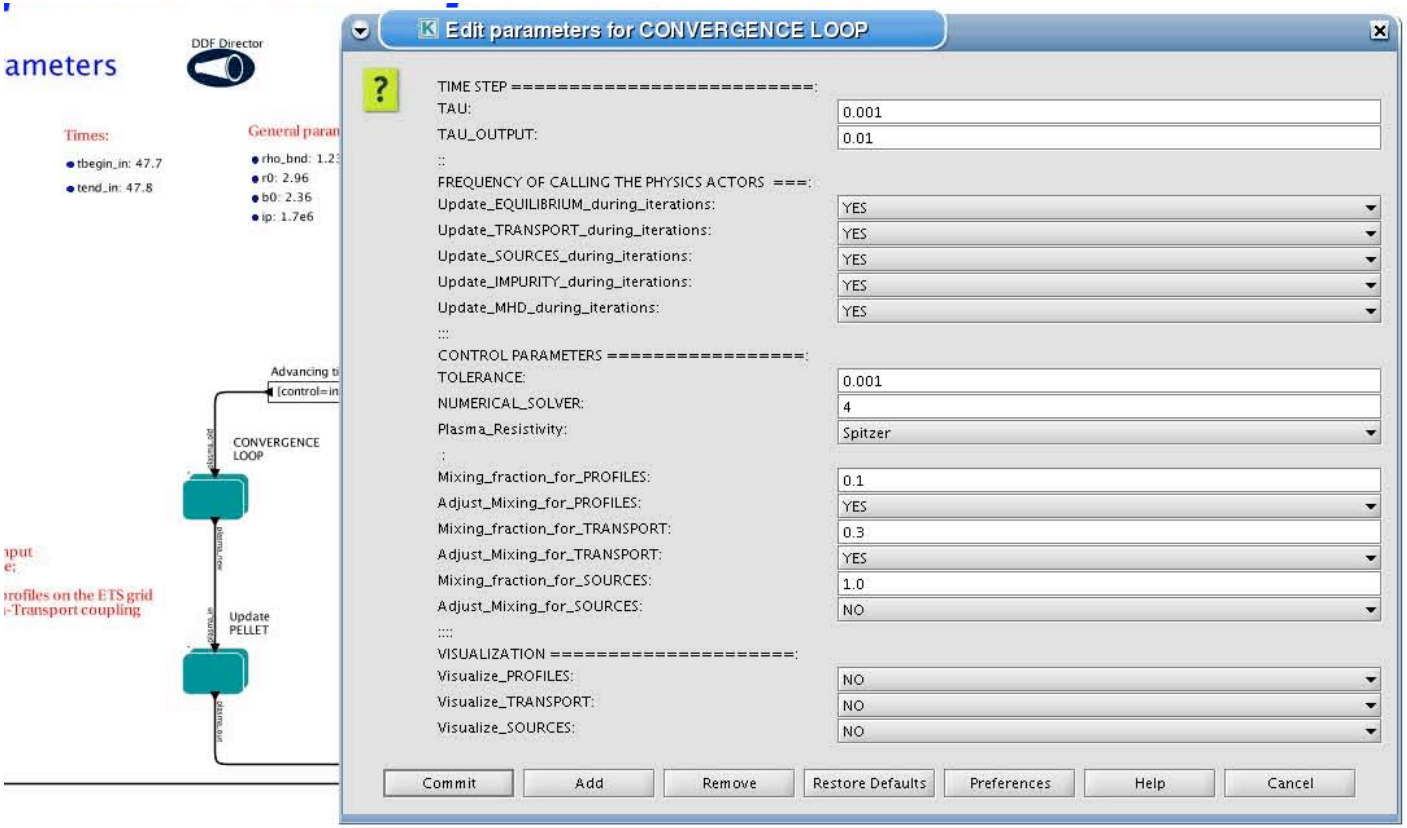


Figure 1
ETS shot: 71827 run: 11 time: 52.3310
Te [keV] „ Ti [keV]
ne [10^19 m-3] „ ni [10^19 m-3]
diff [m^2/s]
Psi
Z
R
q „ jtot [MA/m2]
Zeff [-]
source [-]
1e19
rhotor [m]
rhotor [m]
rhotor [m]

There is a number of visualization scripts which you might activate to follow the iterations.

- ***right click on the box 'CONVERGENCE LOOP'***
- ***select 'Configure actor' to edit settings***
- ***Select 'YES' for needed visualization***
- ***Commit***


ameters
DDF Director
Times:
tbegin_in: 47.7
tend_in: 47.8
General paran
rho_bnd: 1.2
r0: 2.96
b0: 2.36
ip: 1.7e6
Advancing ti
[control=in
CONVERGENCE
LOOP
Update
PELLET
rofiles on the ETS grid
-Transport coupling
Edit parameters for CONVERGENCE LOOP
TIME STEP ==========:
TAU:
0.001
TAU_OUTPUT:
0.01
::
FREQUENCY OF CALLING THE PHYSICS ACTORS ===:
Update_EQUILIBRIUM_during_iterations:
YES
Update_TRANSPORT_during_iterations:
YES
Update_SOURCES_during_iterations:
YES
Update_IMPURITY_during_iterations:
YES
Update_MHD_during_iterations:
YES
:::
CONTROL PARAMETERS ==========:
TOLERANCE:
0.001
NUMERICAL_SOLVER:
4
Plasma_Resistivity:
Spitzer
:
Mixing_fraction_for_PROFILES:
0.1
Adjust_Mixing_for_PROFILES:
YES
Mixing_fraction_for_TRANSPORT:
0.3
Adjust_Mixing_for_TRANSPORT:
YES
Mixing_fraction_for_SOURCES:
1.0
Adjust_Mixing_for_SOURCES:
NO
::::
VISUALIZATION ==========:
Visualize_PROFILES:
NO
Visualize_TRANSPORT:
NO
Visualize_SOURCES:
NO
Commit
Add
Remove
Restore Defaults
Preferences
Help
Cancel