Introduction

Dynamic simulation consists of at least these four steps:

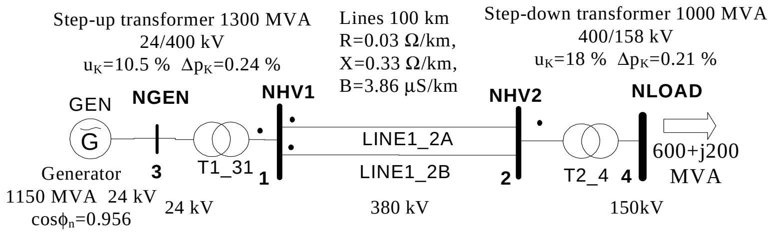
1. Load flow preparation
2. Dynamic model assigning
3. Simulation parameters
4. Simulation and results evaluation.

This short tutorial describes these steps on the simple case study (based on Example 1 from the EUROSTAG – Tutorial).

Some operation instructions are in the User manual which is available from menu **File/Programs documents.**

Example – Input data

The network with necessary load flow data are in the following figure:



Load depends on voltage and frequency according to relations:

$P=P_0(U/U_0)*(f/f_0)$ $Q=Q_0(U/U_0)^2$

It corresponds so called static load model (Application Guide [1] chapter 4.1):

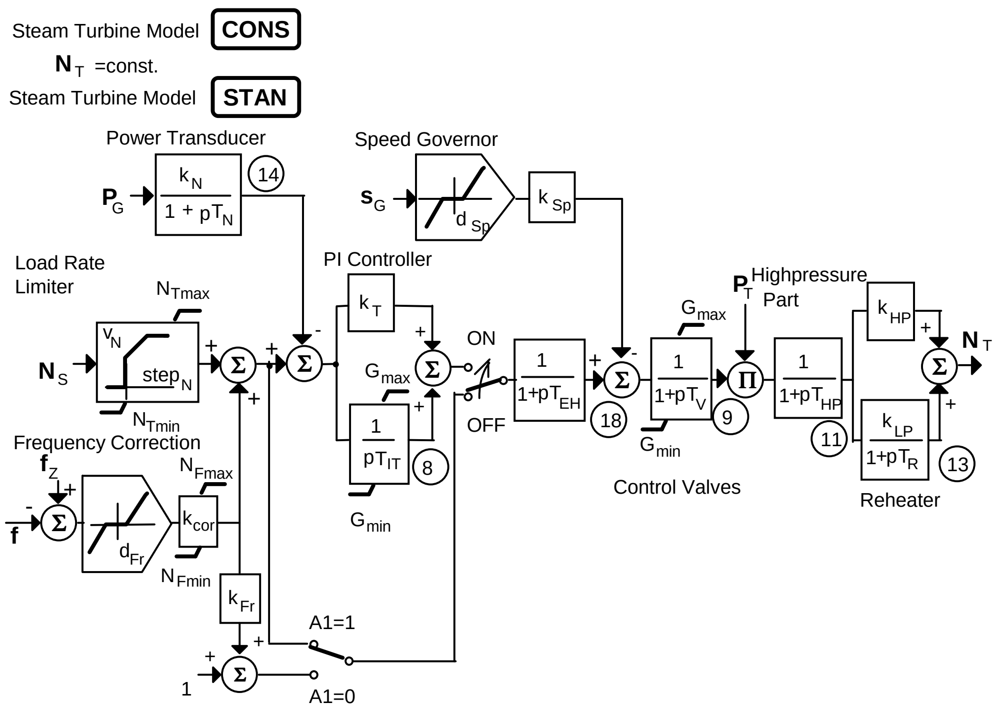
$P_{STAT}=P_0(t)*(1-Ap-Bp+Ap*U+Bp*U^2)*(1+Cp*s_U)/(1-Ap-Bp+Ap*U_0+Bp*U_0^2)$

$Q_{STAT}=Q_0(t)*(1-A_Q-B_Q+A_Q*U_0+B_Q*U_0^2)*(1+C_Q*s_U)/(1-Ap-Bp+Ap*U_0+Bp*U_0^2)$

With parameters: Ap=1, Bp=0, Cp=1 $A_Q=A_0/tg$ $B_Q=1+B_0/tg$ $C_Q=C_0/tg\varphi$ $A_0=0$, $B_0=0$, $C_0=0$

Generator parameters are Un=24 kV, cosφn=0.956, Sn=1150 MVA, $X_d=2.57$, $X_q=2.57$, $X_d'=0.422$, $X_q'=0.662$ $X_d''=0.3$, $T_{d0}'=7.695$ s, $T_{q0}'=0.643$ s, $T_{d0}''=0.061$ s, $T_{q0}''=0.095$ s, $T_m=2H=12.6$ s.

Use constant Efd for excitation system model (CONST) and standard turbine model (STAN) according following figure[1]:



Power control loop is open (switch is in OFF position) with A1=1. Nominal turbine power is Ntn=1000MW. Turbine and speed governor are represented by following parameters: $k_N=Sn/Ntn=1.15$, $d_{Sp}=0$, $k_{Sp}=25$, $T_{EH}=0.01$ s, $T_V=0.01$ s, $T_{HP}=0.01$ s, $T_R=10$ s, $k_{HP}=0.3$, $k_{LP}=0.7$. Other parameters are default.

Following events will be simulated:

1. step change of load of 50+j25 MVA,
2. switching off the line LINE1_2A.

[1] More details on power system modeling is in Application Guide [1] which is available from menu **File/Programs documents**

Load flow data

The best way to prepare new load flow is using of New project and editing its data.

To Open the project:

- Click on the project name NEW in the **Projects** tree.
- Click icon on toolbar or use menu **Project/Open.**
- Confirm **OK** and the MODMAN overwrite working subdirectories VST a VYST by projects files (use the menu **Project/Save** or **Save As** to save proceeding project data**)**.

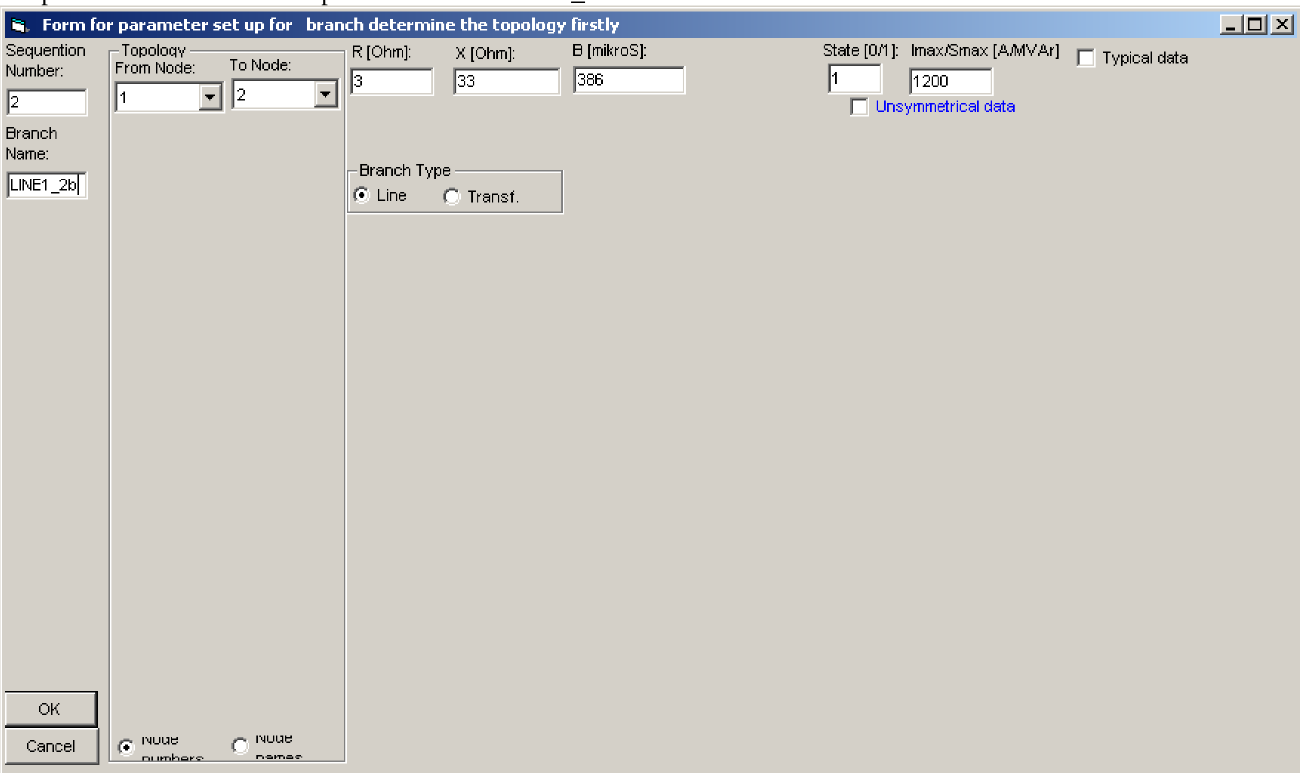
There is only one case named UST_STAV in the NEW project and we use it as a base for our new network creation. Click icon to open Load Flow Editor. Adaptation the two nodes network contains the following steps:

1. to rename the node names UZEL1 and UZEL2 to NHV1 and NHV2
2. to change the reference voltage Uv from 400 kV to 380 kV
3. to change the consumption and generation Pload/Qload and Pgen/Qgen to 0
4. to change the reactive power range Qmin-Qmax to 0
5. to rename the line name VED1_2 to LINE1_2A
6. to change the line parameters R=3, X=33 ,B=386
7. to rename the Unit Name BLOK1 to GEN and Node Name to NGEN
8. to change of nominal power Sn=1150, Ntmax=1000 (nominal turbine power) and unit transformer ratio pt=1

It will be done simply by editing of cells in tables of nodes, branches and units.

To add new parallel line LINE1_2B:

- Use menu **Edit/Add branch**
- Complete form for the similar parameters like LINE1_2A

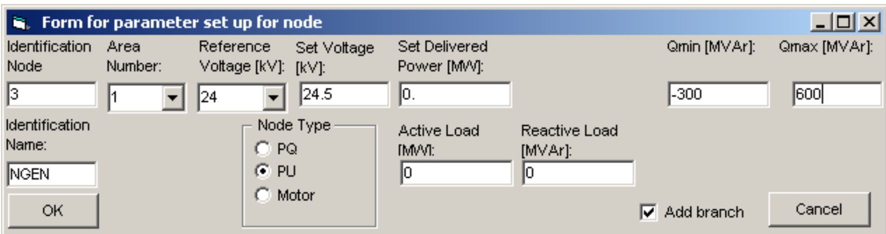


- Click on the **OK** button

Then we add two new nodes NGEN and NLOAD.

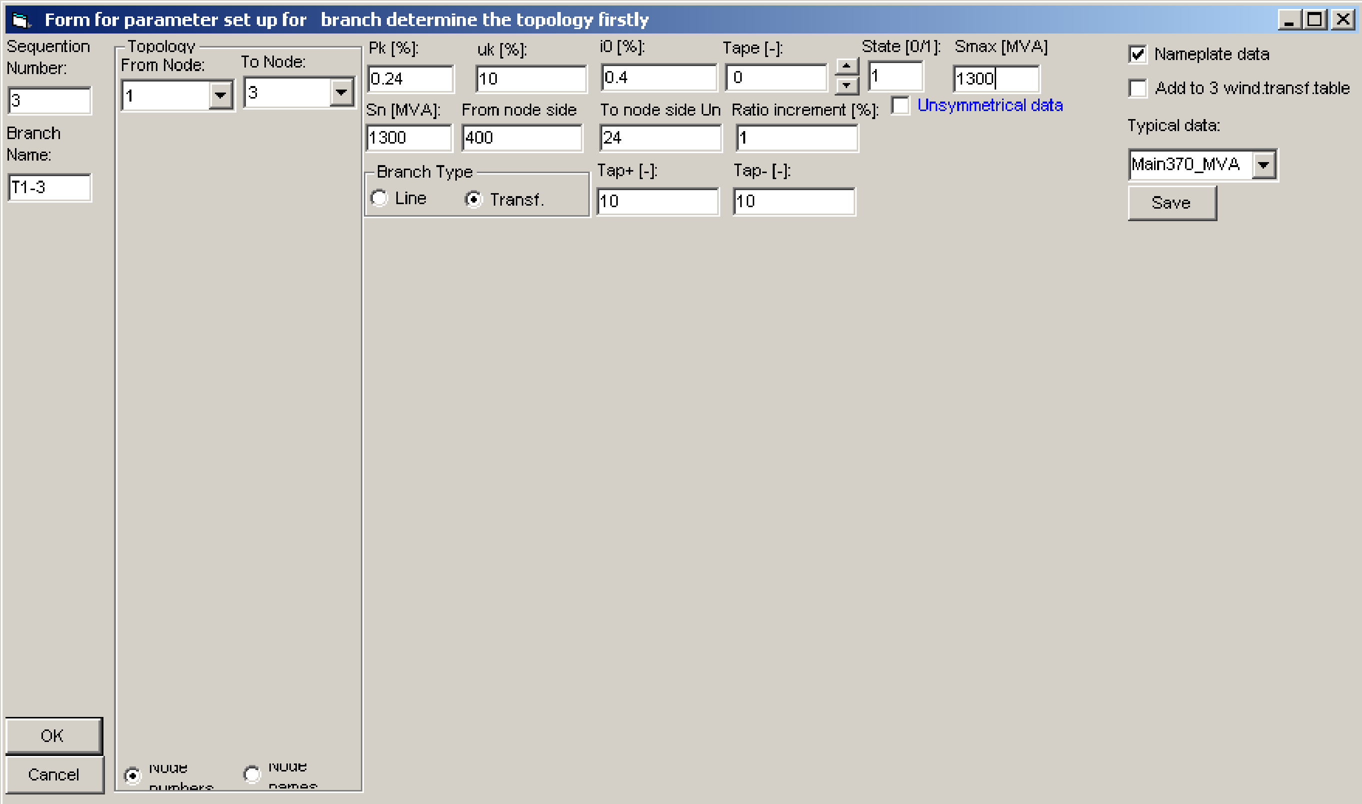
To add new node:

- Click in the table of nodes (the table must be yellow).
- Use menu **Edit/Add node**
- Complete the following form for NGEN node (it is PU type node due to connected generator GEN).



- Click on the **OK** button

- Complete the following form – determine From Node number 1 firstly (in the Topology frame) and then select Transf. for Branch Type.:



- Repeat preceding steps for NLOAD node (it is PQ Node Type with Reference voltage 150 kV and Active/Reactive Load 600/200) and T4_2 branch (determine To Node number 2 firstly and then select Transf. for Branch Type and check Nameplate data, From/To node side Un is 158/400 kV).

To add new parallel line LINE1_2B:

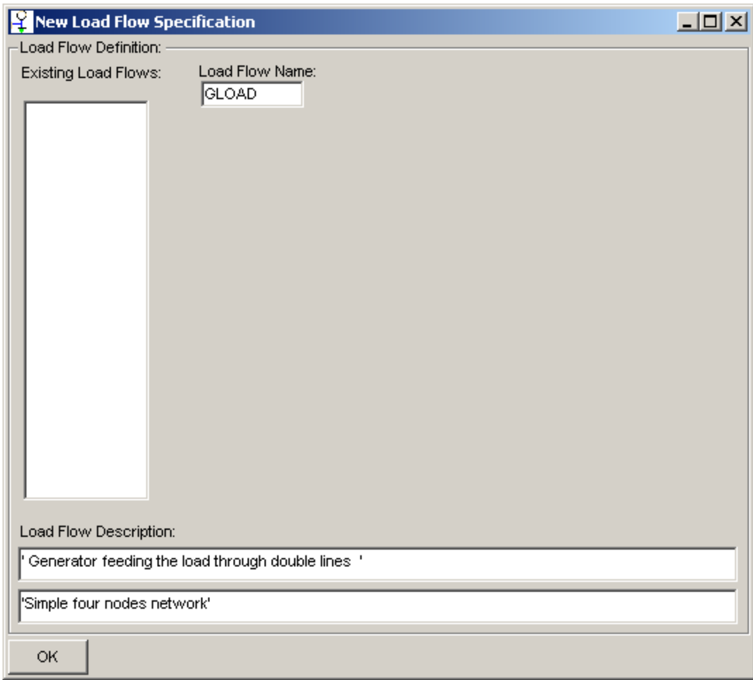
- Use menu **Edit/Add branch**
- Complete form the similar parameters like LINE1_2A NGEN
- Click on the **OK** button

To remove unit EKV_TS:

- Click in the table of units (the table must be yellow) in the row with EKV_TS (▶ must be in the left column)
- Press the Delete key and confirm deleting
- Press **OK** button

Before computing this new load flow it is necessary to perform the following steps:

1. define new Reference node (slack bus) to 3
2. press the **Save** button
3. confirm the topology variation by Yes button
4. complete the form **New Load Flow Specification** - GLOAD is an identificator and Description is adjusted according the new network
5. click OK button
6. confirm the load and units variation by **Yes** buttons

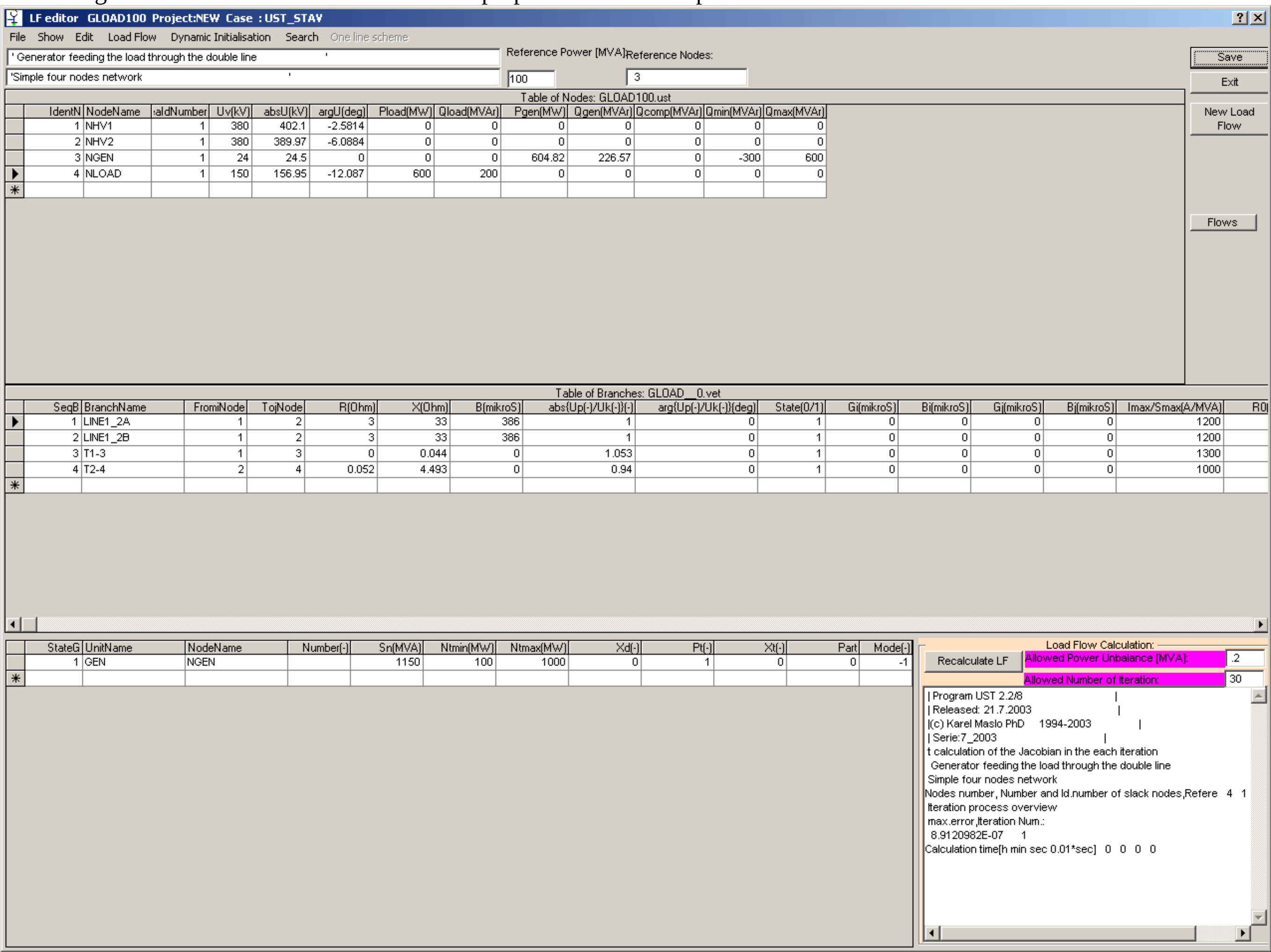


Now it is possible to recalculate the load flow by pressing **Recalculate LF**. After recalculating the first table with nodes is refreshing. Because the voltage in the NLOAD is low (146.3), we increase it.

To change the voltage of the NLOAD:

- Click in the table of branches in line T2-4 (the table must be yellow and ▶ must be in the left column)
- Change the ratio (it is denoted abs{Up/Uk}) of the T2-4 from 0.9993 to 0.94
- Press **Save** button and No for new variant conformation.
- Press **Recalculate LF** button to calculate new load flow.

The voltage is 156.95 kV. The load flow data is prepared for other steps. The load flow data overview is in this form:



Directly from the Load Flow Editor is possible to initialise dynamic models nevertheless no dynamic models are defined. The MODES uses default dynamic models[1] and typical parameters[2].. After selecting **Dynamic initialisation** from the menu three blank text boxes appear in the middle - it means, that starting dynamic models are initialised well and it is possible to carry out next step – specify dynamic models. Press Exit button to return to MODMAN environment.

[1] It is classical model for generators and standard model for turbine
[2] The first set of typical parameters in the global catalogue is default

Dynamic Models

Simply way to assign dynamic data to units is using Unit Models Editor.

- Click on the Unit Models Editor icon on the toolbar.
- Click on the unit GEN
- Press **Add record** button
- Press yellow **Generator** button and select PARK model from the list box
- Press green **Exciter** button and select CONS model from the list box
- Press blue **Turbine** button and switch OFF radio button
- Press **Change all models** to replace default models and confirm all changes

Now we change parameters by simply editing of default parameters. Press again yellow **Generator** button. Then click on the last blank row in the table and press **Add parameters** button. The default set of parameters are copied to this row and it can be edited. Click again in the new row and repair parameters according chapter Example – Input data. Then click on the other row (symbol of pencil disappears) and click back to the editing row. Press the **Change parameters** to exchange default parameters for this new set of parameters S1100.

Generators	Un (kV)	Cosn (-)	Sng (MVA)	Xd (-)	Xq (-)	Xd1 (-)	Xd2 (-)	Xt	Td01 (s)	Td02 (s)	Tq02 (s)	Tm (s)	Xq1 (-)	Tq01 (s)	Coment
S1100	24	0.956	1150	2.57	2.57	0.422	0.3	0	7.695	0.061	0.095	12.6	.0662	0.643	1100MW from Example 1

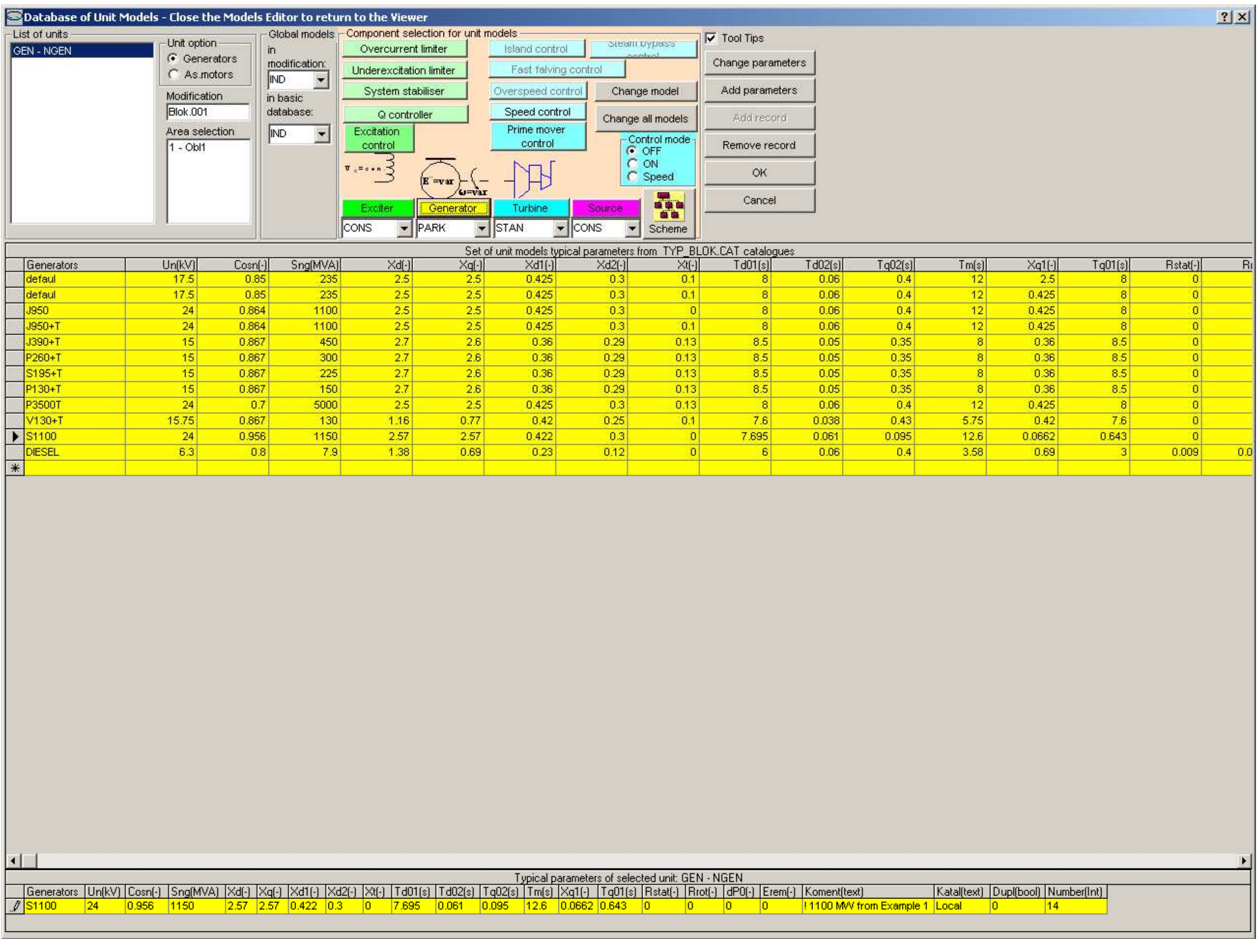
We edit parameters for turbine model. Press again blue **Turbine** button. Click on the default set of parameters and press **Change parameters** button. Further procedure is similar like for generators. Changed parameters are bold in the following table.

Turbines	kN (-)	TV (s)	TI (s)	T_{HP} (s)	T_R (s)	T_{LP} (s)	V_{mn}	V_{mx} (-/s)	V_{Imi} (-/s)	V_{Ima} (-/s)	V_{Csto} (-/s)	V_{Isto} (-/s)	G_{mn} (-)	G_{mx} (-)	K_{LP} (-)	K_{HP} (-)	k_{IV} (-)	Coment
T1000	**1.15**	**.01**	0.2	**.01**	**10**	0.4	-1	0.1	-4	0.67	-4	-4	0	1	**0.7**	**0.3**	2	**Turbine from ..**

The last changes apply to governor. Press again blue **Prime mover control** button and repeat the editing process. Changed parameters are bold in the following table.

Regulator	A1 (-)	A2 (-)	TI (s)	TIB (s)	T_N (s)	T_{EH} (s)	kT (-)	k -	k_{Sp} (-)	k_{Fr} (-)	K_{COR} (-)	k_{Pres}	k_{For} (-)	GEN (-)	vN %/min	step (%)	dFr (%)	dSp (%)	d_{Pres} (%)	dP %	N_{Fmax} (%)	N_{Fmin} (%)	Coment
OPENL	1	0	50	100	1	**0.01**	1.5	1	**25**	0	**0**	1	0.5	1	1	0	0	0	0	0	1.25	1.25	**open loop with speed ....**

The following screen shows the final models selection.



After changing models and editing parameters press **OK**, confirm new modification and saving new parameters into catalogues as well.

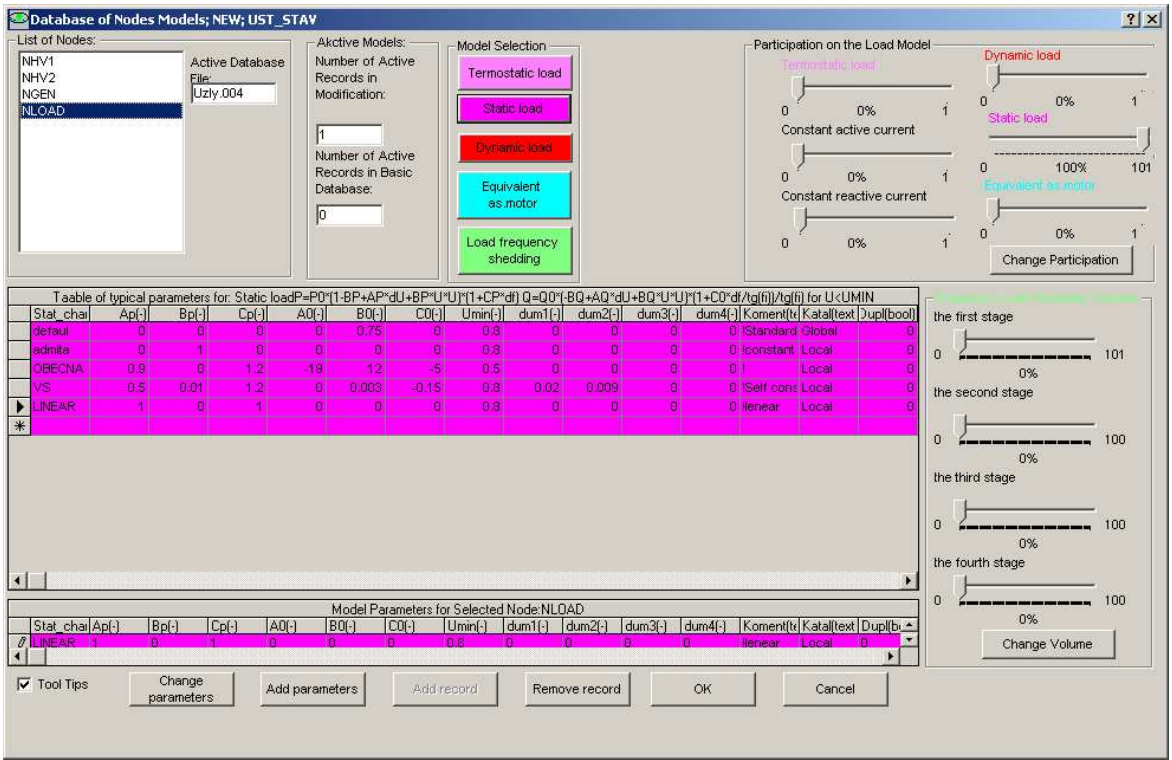
Data for load models are managed by Nodes Models Editor.

- Click on the Nodes Models Editor icon on the toolbar.
- Click on the node NLOAD
- Press **Add record** button
- Press purple **Static load** button
- Move slider for Static load to 100%
- Press **Change participation** button
- Write 1 to the text box Number of active record in Modification.

Now we change parameters by simply editing of default parameters. Click on the last blank row in the table and press **Add parameters** button. The default set of parameters are copied to this row and it can be edited. Click again in the new row and repair parameters according chapter Example – Input data. Then click on the other row (symbol of pencil disappears) and click back to the editing row. Press the **Change parameters** to exchange default parameters for this new set of parameters LINEAR displayed in the following table:

Regulator	AP (-)	BP (-)	CP (-)	A0 (-)	B0 (-)	C0 (-)					Coment
LINEAR	1	0	1	0	0	0					Linear P dependency on U /f for Example 1

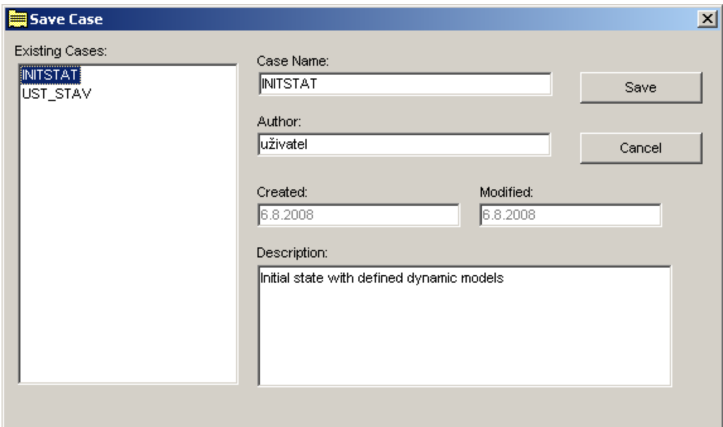
The following screen shows the final models selection.



Press **OK**, confirm new modification and saving new parameters into catalogues as well.

Dynamic models are ready now and it is possible to run the simulation by clicking on icon in the toolbar. Standard graphic occurs on the display (Active and reactive power output PG and QG, speed deviation SG and terminal voltage UG of the first generator) and it shows a steady state. Press E key to exit

We can save our work in this phase like case. Click on the menu **Cases/Save as** and fill the following form.



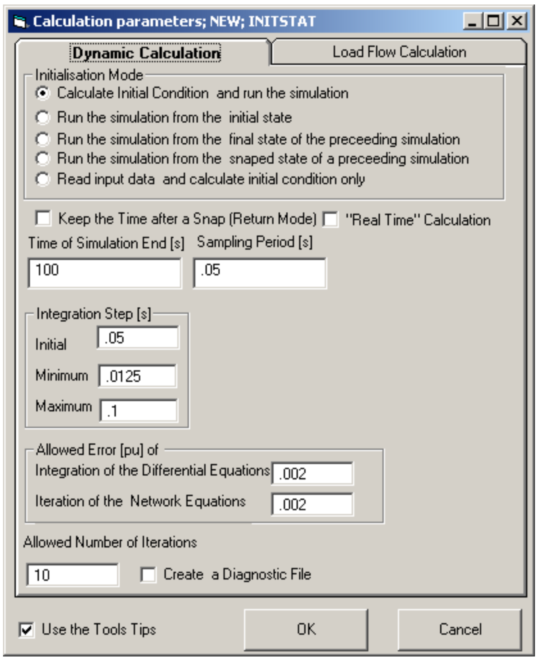
Then press the **Save** button. Now we can continue the solution.

Simulation parameters

Simulation parameters contains especially:

- Simulation time and sampling periods
- Definition of scenario - sequence of simulation events
- Definition output variables - displayed during simulation
- Determination of output files for post processing.

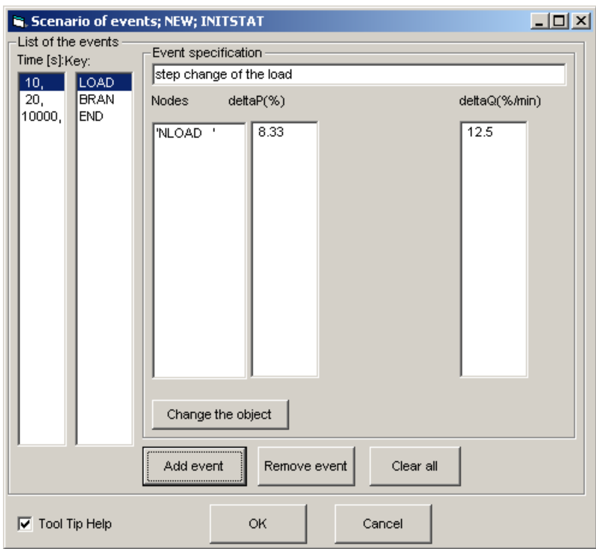
Click on the menu **Modify/Control** to change simulation parameters and fill the following form:



Then press the **OK** button and confirm Yes to create new variant.

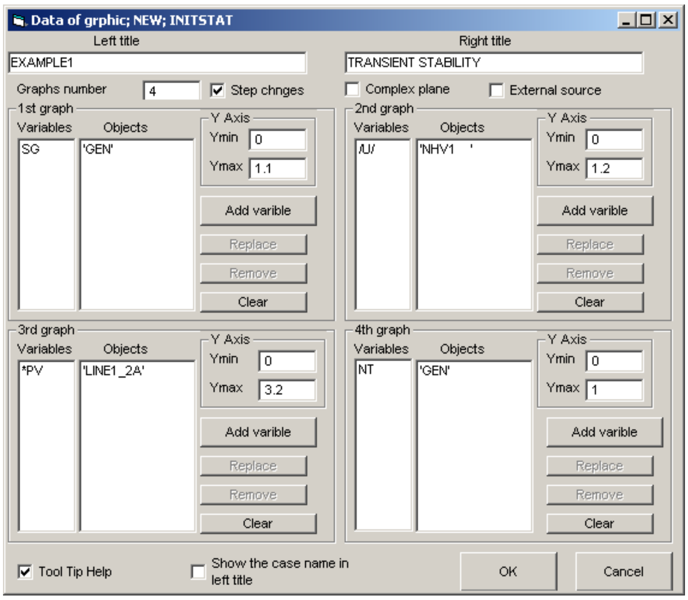
Simply way to define events is using Scenario dialog box.

- Click on the first icon  in the third groups on the toolbar.
- Click on **Add event** button
- Write time 10 s
- Select **Nodes** in Object types frame
- Click on **Add object** button
- Select NLOAD from **Node** combo box
- Write deltaP=50*100/600=8.33 % and deltaQ=25*100/200=12.5 % into **Parameter specification** frame
- Press **Add** and **Cancel** buttons twice
- Click on **Add event** button once more
- Write time 20 s
- Select **Branches** in Object types frame
- Click on **Add object** button
- Select LINE1_2A from **Line** combo box
- Press **Add** and **Cancel** buttons twice,
  so that scenario dialog looks like:
- Press **OK** and confirm creation of new variant.



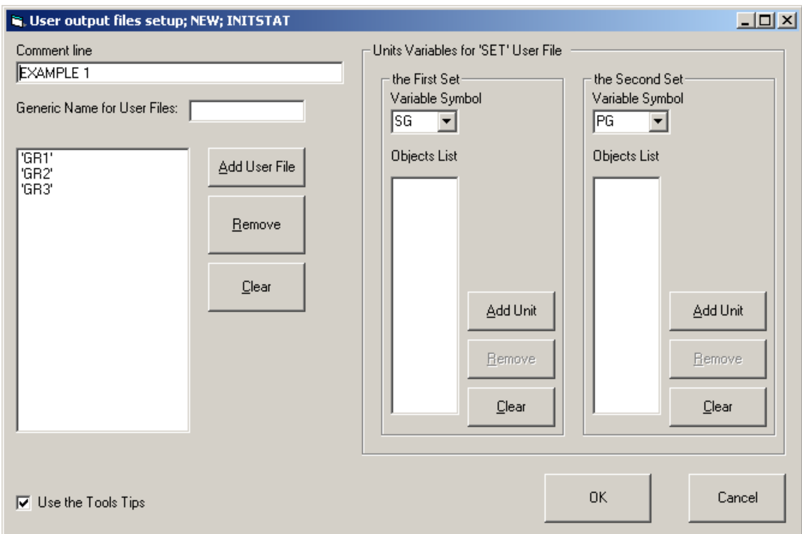
Simply way to define output variables is using Graphic dialog box.

- Click on the second icon  in the third groups on the toolbar.
- Write Example 1 to the **Left title** text box
- Write 4 to the Graphs number text box
- Click on **Clear** button in the 1st graph frame
- Click on **Add variable** button
- Select SG from **Variables** combo box
- Press **Add** and **Cancel** buttons
- Write 0 and 1.2 to the text boxes **Ymin** and **Ymax** in the 2nd graph frame
- Click on **Clear** button in the 2nd graph frame
- Click on **Add variable** button
- Select **Nodes** in Object selection frame
- Select NHV1 from **Node** combo box
- Select /U/ from **Variables** combo box
- Press **Add** and **Cancel** buttons
- Write 0 and 3.2 to the text boxes **Ymin** and **Ymax** in the 3rd graph frame
- Click on **Clear** button in the 3rd graph frame
- Click on **Add variable** button
- Select **Branches** in Object selection frame
- Press **Add** and **Cancel** buttons
- Similarly like SG Add variable NT (turbine output) into the 4st graph, so that graphic dialog looks like:
- Press **OK** and confirm creation of new variant.



It is necessary to define output files for investigation of simulation time courses after finishing of calculation. Click on the menu **Modify/User File** to define these files and fill the following form:

- Write Example 1 to the **Comment line** text box
- Delete **Generic Name for User Files** text box
- Click on **Add User file** button
- Select **Variables from display** tap
- Select **Variables from the first graph** radio button and press **Add** button
- Select **Variables from the second graph** radio button
- Press **Add** and **Cancel** buttons, so that scenario dialog looks like:



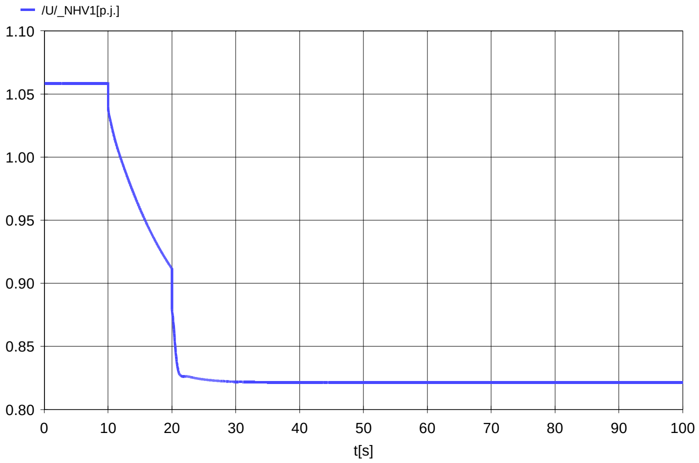
Then press the **OK** button and confirm Yes to create new variant.

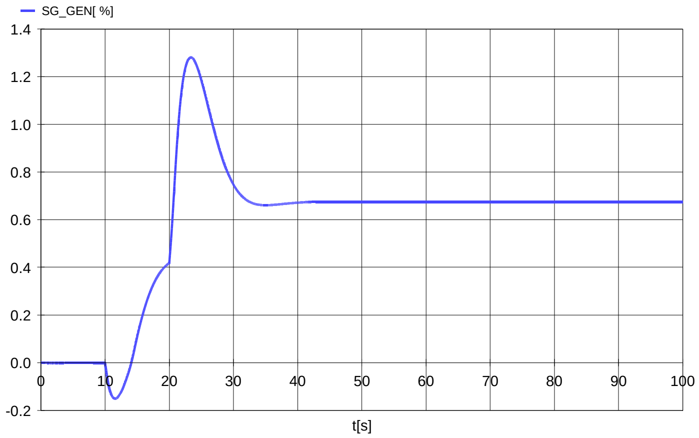
Simulation and results evaluation

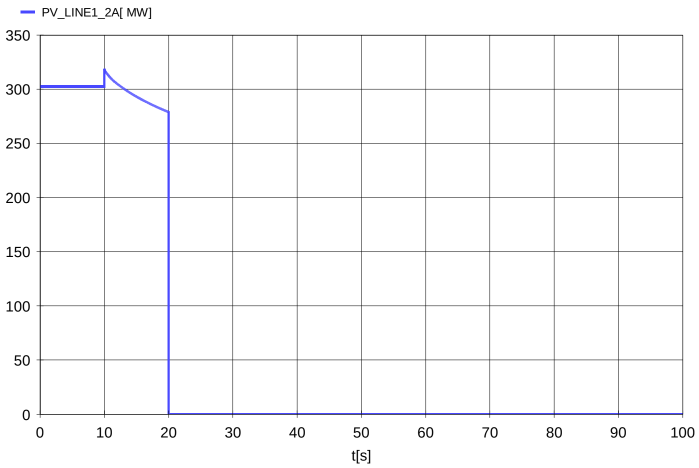
Now we can repeat the simulation by pressing the on icon in the toolbar. You can see the system response on events determined by scenario directly on display. Because the response of the system and calculation result are satisfied we can save the calculation like case named like LINEOUT.

It is possible to show predefined variables time course after calculation. Check **As graph** check box in the **Results** menu and then click on the icon on the toolbar. Four icons appear on the toolbar. You can examine of time course by clicking on these icons. The following figures show the time courses.

The voltage /U/ decreases after load step change and line outage. Load decreases consequently due to regulation effect (especially due to voltage dependency). Frequency deviation increases due to power excess in the island.







Now when we have finished work we can save it like a project. Click on the menu **Projects/Save as** and define the name (e.g. Tutorial) and description.

Reference

[1] MODES 2.2/2 Application Guide 3rd Edition 10/1995