Rock Flow Dynamics

Reservoir Simulator

tNavigator
version 3.2

User Manual

Moscow, 2011

Reservoir Simulator tNavigator (version 3.2). User Manual. — Moscow, 2011. — 852 pp.

Authors: Bogachev K.Yu., Sorokina M.M., Mikhaleva M.Yu.
Editors: Prohorenko M.I., Egorova A.A.

Содержание

1   Introduction

Simulator tNavigator can be used for numerical solution of three phase three (or multi) component filtration problems (black-oil, compositional or thermal model). Simulator uses finite volume approximation on rectangular block centered mesh with respect to space. For approximation with respect to time Fully Implicit method is used.

Hydrodynamic simulator tNavigator is recommended for calculation of oil and gas field development plan projects.

For best adaptation of user experience the keyword notations are chosen to be close as much as possible to the most common simulators:

- Eclipse (c) Schlumberger,

- IMEX (c) Computer Modelling Group Ltd,

- STARS (c) Computer Modelling Group Ltd,

- MORE (c) Roxar.

tNavigator reads keyword notations of these simulators and converts them into its inner data notations. This User Manual describes all keywords which can be used in tNavigator:

- tNavigator keywords;

- Eclipse 100 keywords;

- Eclipse 300 keywords;

- CMG IMEX keywords;

- CMG GEM keywords;

- CMG STARS keywords.

This description pointed out if there are parameters of the keyword which are ignored by tNavigator or which use is different from Eclipse (CMG).

Eclipse compatible keywords are red. For example: TABDIMS (see 6.1.13). Index of Eclipse compatible keywords – 11.

CMG compatible keywords are pink. For example: TEMR (see 8.4.4). Index of CMG compatible keywords – 12.

MORE compatible keywords are green. For example: IDATe (see 9.1.5). Index of MORE compatible keywords – 14.

The keywords which RFD uses only in tNavigator are blue. For example: REACCONC (see 6.8.46). Index of RFD compatible keywords – 13.

tNavigator simulator is subject to future development. Any feedback is appreciated.

2 Physical model

Simulator uses standard three phase three component isothermic black-oil model and compositional model. The description of thermal compositional model with chemical reactions is in the section 3.

2.1 Differential equations for black-oil model

Standard black-oil equations with standard assumptions:

$$\frac{\partial}{\partial t}(\phi N_c) = \operatorname{div}\sum_{P=O,W,G} x_{c,P}\xi_P\left(\boldsymbol{k}\frac{k_{rP}}{\mu_P}(\nabla p_P - \gamma_P\nabla D)\right) + q_c, \quad c = 1,\dots,n_c \quad (2.1)$$

$$p_O - p_G = P_{cOG}, \tag{2.2}$$

$$p_O - p_W = P_{cOW}, \tag{2.3}$$

$$S_W + S_O + S_G = 1. \tag{2.4}$$

Here functions:

- $N_c = N_c(t,x,y,z)$ (**unknown**) – $c = 1,\dots,n_c$ overall molar density of any component. For black oil model components are water, oil and gas, and

$$N_w = \xi_{W,SC}\frac{S_W}{B_W}; \quad N_o = \xi_{O,SC}\left(\frac{S_O}{B_O} + R_{O,G}\frac{S_G}{B_G}\right); \quad N_g = \xi_{G,SC}\left(\frac{S_G}{B_G} + R_{G,O}\frac{S_O}{B_O}\right)$$

- $S_P = S_P(t,x,y,z)$ (**unknown**) – phase P, $P = O,G,W$ saturation,

- $R_{G,O} = R_{G,O}(p_O)$ – solubility of gas component into oil phase (**known** function) (see 2.13),

- $R_{O,G} = R_{O,G}(p_O)$ – vaporisation of oil component into gas phase (**known** function) (see 2.14),

- $B_P = B_P(p_P)$ – phase formation volume factor (**known** function) (see 2.9),

- $\phi_P = \phi(p_P,x,y,z)$ – porosity (**known** functions) (see 2.5),

- $p_W = p_W(t,x,y,z)$ (**unknown**) – water phase pressure,

- $p_O = p_O(t,x,y,z)$ (**unknown**) – oil phase pressure,

- $p_G = p_G(t,x,y,z)$ (**unknown**) – gas phase pressure,

- $x_{c,P} = x_{c,P}(p,\boldsymbol{N})$ (**known** function) – moles of component c per mole of phase P,

- $\xi_P = \xi_P(p,\boldsymbol{N})$ – phase molar density (**known** function), see section 2.10,

- $\boldsymbol{k} = \boldsymbol{k}(p_W,p_O,p_G,x,y,z)$ – permeability tensor (**known** function) (see 2.4),

- $k_{rP} = k_{rP}(S_W, S_G)$ – phase relative permeability (**known** function) (see 2.6),

- $\mu_P = \mu_P(p_P)$ – phase viscosity (**known** function) (see 2.8),

- $\gamma_P = \rho_P g$ – vertical pressure gradient (**known** relation),

- $D = D(x, y, z)$ – vertical depth vector (up-down oriented) (**known** coordinate functions),

- $\rho_P = \rho_P(p_P)$ – phase mass density (**known** function) (see 2.11),

- $P_{cOG} = P_{cOG}(S_G)$ – oil-gas capillary pressure (**known** function) (see 2.12.1),

- $P_{cOW} = P_{cOW}(S_W)$ – oil-water capillary pressure (**known** function) (see 2.12.2),

- $q_c = q_c(p, N, t, x, y, z)$ – source of component (**known** function) (see 2.15.1)

and constant(s):

- $g = \text{const}$ – **known** constant

Let us define the ways of known data input.

2.2 Boundary conditions

The standard constant pressure (Dirichlet)

$$p_P = \text{const}_P \tag{2.5}$$

boundary conditions or standard no flow (Neumann)

$$\frac{\partial p_P}{\partial N} = \Big(\lambda_P(\nabla p_P - \gamma_P \nabla D), \boldsymbol{n}\Big) = 0 \tag{2.6}$$

boundary conditions are used on outer reservoir boundary. Here $\lambda_P = \boldsymbol{k}\dfrac{k_{rP}}{B_P \mu_P}$.

2.3 Initial conditions

Initial conditions may either all known values for p_P, S_P (from previous run of the model) or values for p_P, S_P may be computed from hydrostatic equilibrium conditions:

$$\text{div} \sum_{P=O,W,G} x_{c,P} \xi_P \Big(\boldsymbol{k} \frac{k_{rP}}{\mu_P} (\nabla p_P - \gamma_P \nabla D) \Big) = 0 \tag{2.7}$$

$$p_O - p_G = P_{cOG} \tag{2.8}$$

$$p_O - p_W = P_{cOW} \tag{2.9}$$

$$S_W + S_O + S_G = 1 \tag{2.10}$$

with boundary conditions from 2.2.

EQUIL (see 6.9.1) specifies initial values for p_P, S_P (PRESSURE (see 6.9.7), SGAS (see 6.9.10), SWAT (see 6.9.9), SOIL (see 6.9.11), SWATINIT (see 6.6.30)).

2.4 Permeability tensor

Absolute permeability tensor $k = k(p_W, p_O, p_G, x, y, z)$ is user input data array function defined in all reservoir points. PERMX / PERMY / PERMZ (see 6.2.7) Usually it is diagonal tensor but in the current version anisotropic tesor is supported too (PERMXY / PERMYZ / PERMXZ (see 6.2.8)). On default dependence of k on pressure is omitted. One can specify this dependence using the keyword ROCKTAB (see 6.2.10).

2.5 Porosity

Porosity $\phi = \phi(p, x, y, z)$ is user input data function defined in all reservoir points. Usually it is represented in the following form:

$$\phi(p, x, y, z) = \psi(x, y, z)\phi(x, y, z)(1 + c(p - p_{\text{ref}}) + c^2(p - p_{\text{ref}})^2/2)$$

where

- $\psi(x, y, z)$ – net to gross (user input data array defined in all reservoir points, NTG (see 6.2.18) or DZNET (see 6.2.19))

- $\phi(x, y, z)$ – porosity at pressure p_{ref} (user input data array defined in all reservoir points, PORO (see 6.2.17))

- c – compressibility (user input data, ROCK (see 6.5.12) or ROCKTAB (see 6.2.10))

- p_{ref} – reference pressure (user input data, ROCK (see 6.5.12))

2.6 Phase relative permeability

Phase relative permeability $k_{rP} = k_{rP}(S_W, S_G)$ are defined by experimental data. The usual assumptions are:

$$
\begin{aligned}
k_{rW} &= k_{rW}(S_W) & (2.11) \\
k_{rG} &= k_{rG}(S_G) & (2.12) \\
k_{rO} &= k_{rO}(S_W, S_G) & (2.13)
\end{aligned}
$$

User specifies two sets of relative permeabilities function pairs:

- $k_{rWO} = k_{rWO}(S_W)$, $k_{rOW} = k_{rOW}(S_W)$ – for water-oil two phase system,

- $k_{rGO} = k_{rGO}(S_G)$, $k_{rOG} = k_{rOG}(S_G)$ – for gas-oil two phase system

(SWOF (see 6.6.1) for first pair, SGOF (see 6.6.2) for second pair). These functions may be obtained by laboratory measurements or may be approximated by analytical functions based on the following user input data:

- S_{WC} – connate water saturation

- S_{OrW} – residual oil saturation to waterflooding

- S_{Gr} – critical gas saturation

- S_{OrG} – residual oil saturation to gasflooding

- k_{rWrO} – water relative permeability at residual oil and $S_G = 0$

- k_{rOcW} – oil relative permeability at S_{WC} and S_{Gr}

- k_{rGcW} – gas relative permeability at S_{WC} and $S_O = 0$

- n_W, n_{OW}, n_G, n_{OG} – constant function parameters

Then we can define analytical approximations for the listed above functions:

$$k_{rWO}(S_W) = \begin{cases} 0 & \text{if } S_W \le S_{WC} \\ k_{rWrO} \left(\dfrac{S_W - S_{WC}}{1 - S_{WC} - S_{OrW}} \right)^{n_W} & \text{otherwise} \end{cases}$$

$$k_{rOW}(S_W) = \begin{cases} 0 & \text{if } 1 - S_W - S_{OrW} < 0 \\ k_{rOcW} \left(\dfrac{1 - S_W - S_{OrW}}{1 - S_{WC} - S_{OrW}} \right)^{n_{OW}} & \text{otherwise} \end{cases}$$

$$k_{rGO}(S_G) = \begin{cases} 0 & \text{if } S_G < S_{Gr} \\ k_{rGcW} \left(\dfrac{S_G - S_{Gr}}{1 - S_{Gr} - S_{wr}} \right)^{n_G} & \text{otherwise} \end{cases}$$

$$k_{rOG}(S_G) = \begin{cases} 0 & \text{if } 1 - S_G - S_{Wc} - S_{OrG} < 0 \\ k_{rOcW} \left(\dfrac{1 - S_G - S_{Wc} - S_{OrG}}{1 - S_{Wc} - S_{OrG}} \right)^{n_{OG}} & \text{otherwise} \end{cases}$$

Then we define:

$$\begin{aligned} k_{rW}(S_W) &= k_{rWO}(S_W), & (2.14) \\ k_{rG}(S_G) &= k_{rGO}(S_G). & (2.15) \end{aligned}$$

At present two options for k_{rO} calculations are available: first and second Stone models. By default the first Stone model is used.

2.6.1 The first Stone's model

Let us define the following constants:

- S_{Wc} – connate water saturation, which is the minimal admissible value of S_W for water-oil two phase system

- $k_{rOcW} = k_{rOW}(S_{Wc})$ – relative permeability to oil at connate water

- S_{Wr} – residual water saturation, which is the largest value of S_W, where $k_{rW}(S_W) = k_{rWO}(S_W) = 0$

- S_{OrW} – water saturation at residual oil, which is the largest value of S_W, where $k_{rOW}(S_W) = 0$

- S_{OrG} – gas saturation at residual oil, which is the largest value of S_G, where $k_{rOG}(S_G) = 0$.

Then

$$\alpha(S_G) = 1 - \frac{S_G}{1 - S_{Wc} - S_{OrG}},$$

$$S_{om}(S_G) = \alpha(S_G)S_{OrW} + (1 - \alpha(S_G))S_{OrG},$$

$$S_O^*(S_O, S_G) = \begin{cases} \dfrac{S_O - S_{om}(S_G)}{1 - S_{Wc} - S_{om}(S_G)}, & \text{if } S_O \geq S_{om}(S_G) \\ 0, & \text{otherwise} \end{cases},$$

$$S_W^*(S_W, S_G) = \begin{cases} \dfrac{S_W - S_{Wc}}{1 - S_{Wc} - S_{om}(S_G)}, & \text{if } S_W \geq S_{Wc} \\ 0, & \text{otherwise} \end{cases},$$

$$S_G^*(S_G) = \frac{S_G}{1 - S_{Wc} - S_{om}(S_G)}, \tag{2.16}$$

and

$$k_{rO} = k_{rO}(S_O, S_W, S_G) = \frac{S_O^*(S_O, S_G)}{k_{rOcW}} \frac{k_{rOW}(S_W)}{1 - S_W^*(S_W, S_G)} \frac{k_{rOG}(S_G)}{1 - S_G^*(S_G)}. \tag{2.17}$$

2.6.2 The second Stone's model

Using the same constants as defined in the previous paragraph:

- S_{Wc} – connate water saturation, which is the minimal admissible value of S_W for water-oil two phase system

- $k_{rOcW} = k_{rOW}(S_{Wc})$ – relative permeability to oil at connate water

we get the following expression for oil relative permeability in case of Stone 2:

$$k_{rO}(S_O, S_W, S_G) = k_{rOcW} \left(\frac{k_{rOW}(S_W)}{k_{rOcW}} + k_{rW}(S_W) \right) \left(\frac{k_{rOG}(S_G)}{k_{rOcW}} + k_{rG}(S_G) \right) \tag{2.18}$$
$$- k_{rOcW}(k_{rW}(S_W) + k_{rG}(S_G))$$

2.6.3 End-point scaling, two-point method

The keyword ENDSCALE (see 6.6.11) indicates that end-point scaling of relative permeabilities and capillary pressures will be used. Table end-points can then be entered cell by cell (SWL (see 6.6.14), SWCR (see 6.6.16), SWU (see 6.6.20), KRW (see 6.6.26), PCW (see 6.6.28)) or with respect to depth (ENPTVD (see 6.6.22), ENKRVD (see 6.6.23), ENPCVD (see 6.6.24)).

If end-point scaling option is selected (ENDSCALE (see 6.6.11)), saturations and relative permeabilities are renormalilzed according to formulas below.

Saturations scaling

First introduce notations

- S_W, S_G are block water and gas saturations,

- S_{Wcr}, S_{Gcr} are critical water and gas saturations, i.e. maximal water (gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)), for which $k_{rW} = 0$ ($k_{rG} = 0$),

  If TOLCRIT (see 6.6.8) isn't specified the critical water saturation S_{Wcr} is equal to S_W in the last table entry (SWOF (see 6.6.1), SWFN (see 6.6.4)) for k_{rW}, for which $k_{rW} \leq 1.0 * 10^{-6}$ – in e100 models ($k_{rW} \leq 1.0 * 10^{-20}$ – in e300 models) (finding the last zero relative permeability value while accounting for machine zero).

  If TOLCRIT (see 6.6.8) is specified, the critical water saturation is equal to S_W in the last table entry, for $k_{rW} \leq TOLCRIT$ (analogously S_{Gcr}, S_{OWcr}, S_{OGcr}).

- S_{Wmax}, S_{Gmax} are maximal values of water (and gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)),

- SWCR (see 6.6.16), SGCR (see 6.6.17) are user defined values of critical water (and gas) saturation in current block,

- SWU (see 6.6.20), SGU (see 6.6.21) are user defined values of maximal water (and gas) saturation in current block,

- $\tilde{S}_W$, $\tilde{S}_G$ are scaled block water and gas saturations,

- k_{rWO}, k_{rGO} are water and gas relative permeabilities defined by SWOF (see 6.6.1) (SGOF (see 6.6.2)) tables,

- $k_{rW\ max}(table)$, $k_{rG\ max}(table)$ are maximum entry of water (gas) relative permeability in SWOF (see 6.6.1) (SGOF (see 6.6.2)) table.

$$\tilde{S}_W = S_{Wcr} + \frac{(S_W - SWCR)(S_{Wmax} - S_{Wcr})}{SWU - SWCR}$$
$$\tilde{S}_G = S_{Gcr} + \frac{(S_G - SGCR)(S_{Gmax} - S_{Gcr})}{SGU - SGCR} \tag{2.19}$$

The keyword TZONE (see 6.6.12) controls the transition zone option. If the parameter is set – true to a phase, then the critical saturations for that phase will be modified to be the initial immobile saturation in regions where the saturation is below the input critical value.

1. the parameter is set true to oil phase – SOWCR (see 6.6.18) will be modified for oil-water runs or oil-water-miscible gas runs, SOGCR (see 6.6.19) will be modified only for oil-gas runs;

2. the parameter is set true to water phase, SWCR (see 6.6.16) will be modified;

3. the parameter is set true to gas phase, SGCR (see 6.6.17) will be modified for gas-water runs and oil-gas runs.

Appropriate relative permeabilities are caculated as

$$k_{rW}(S_W) = \begin{cases} 0 & S_W \leq SWCR \\ k_{rWO}(\tilde{S}_W) & SWCR < S_W < SWU \\ k_{rW\ max}(table) & S_W \geq SWU \end{cases} \tag{2.20}$$

$$k_{rG}(S_G) = \begin{cases} 0 & S_G \leq SGCR \\ k_{rGO}(\tilde{S}_G) & SGCR < S_G < SGU \\ k_{rG\ max}(table) & S_G \geq SGU \end{cases} \tag{2.21}$$

Oil relative permeabilties are calculated in analogous way.

Relative permeabilities scaling
If at least one of the following arrays KRO (see 6.6.25), KRW (see 6.6.26), KRG (see 6.6.27), KRORW (see 6.6.25), KRORG (see 6.6.25), KRWR (see 6.6.26), KRGR (see 6.6.27) is set, relative permeabilities are additionaly scaled according to following formulas. Here SR and SPCR are defined analogously to 3-point scaling case.

Water:

If only KRW is set
$$k_{rW}^{scaled}(S_W) = k_{rW}(S_W) \cdot \frac{KRW}{k_{rW\ max}(table)} \tag{2.22}$$

If both KRW and KRWR are set

$$S_W <= SR \quad k_{rW}^{scaled}(S_W) = k_{rW}(S_W) \cdot \frac{KRWR}{k_{rW}(SR)} \tag{2.23}$$

$$S_W > SR \quad k_{rW}^{scaled}(S_W) = KRWR + \frac{KRW - KRWR}{k_{rW\ max}(table) - k_{rW}(SR)} \cdot (k_{rW}(S_W) - k_{rW}(SR))$$

Gas:

If only KRG is set

$$k_{rG}^{scaled}(S_G) = k_{rG}(S_G) \cdot \frac{KRG}{k_{rG\ max}(table)} \qquad (2.24)$$

If both KRG and KRGR are set

$$S_G <= SR \qquad k_{rG}^{scaled}(S_G) = k_{rG}(S_G) \cdot \frac{KRGR}{k_{rG}(SR)} \qquad (2.25)$$

$$S_G > SR \qquad k_{rG}^{scaled}(S_G) = KRGR + \frac{KRG - KRGR}{k_{rG\ max}(table) - k_{rG}(SR)} \cdot (k_{rG}(S_G) - k_{rG}(SR))$$

Oil: here "P" stands to water or gas phase

If only KRO is set

$$k_{rOP}^{scaled}(S_P) = k_{rOP}(S_P) \cdot \frac{KRO}{k_{rOP\ max}(table)} \qquad (2.26)$$

If both KRO and KRORP are set

$$S_P <= SPCR \qquad k_{rOP}^{scaled}(S_P) = k_{rOP}(S_P) \cdot \frac{KRORP}{k_{rOP}(SPCR)} \qquad (2.27)$$

$$S_P > SPCR \qquad k_{rOP}^{scaled}(S_P) = KRORP +$$
$$+ \frac{KRO - KRORP}{k_{rOP\ max}(table) - k_{rOP}(SPCR)} \cdot (k_{rOP}(S_P) - k_{rOP}(SPCR))$$

2.6.4 End-point scaling, three-point method

If in addition to ENDSCALE (see 6.6.11) three point scaling method for relative permeabilities is selecited (SCALECRS (see 6.6.13)), phase permeabilities are recalculated in the following manner.

Saturations scaling
As for two-point case, first introduce notations

- S_W, S_G are block water and gas saturations,

- $S_{W\ co}$, $S_{G\ co}$ are connate water and gas saturations, i.e. minimal water (gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)),

- S_{Wcr}, S_{Gcr} are critical water and gas saturations, i.e. maximal water (gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)), for which $k_{rW} = 0$ ($k_{rG} = 0$),

- S_{OWcr}, S_{OGcr} are critical oil-to-water and oil-to-gas saturations, i.e. maximal oil saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)), for which the oil relative permeability is zero: $k_{rOW} = 0$ ($k_{rOG} = 0$),

- S_{Wmax}, S_{Gmax} are maximal values of water (and gas) saturation in **SWOF** (see 6.6.1) (**SGOF** (see 6.6.2)),

- **SWL** (see 6.6.14), **SGL** (see 6.6.15) are user defined values of connate water (and gas) saturation in current block,

- **SWCR** (see 6.6.16), **SGCR** (see 6.6.17) are user defined values of critical water (and gas) saturation in current block,

- **SOWCR** (see 6.6.18), **SOGCR** (see 6.6.19) are user defined values of critical oil-to-water (and oil-to-gas) saturation in current block,

- **SWU** (see 6.6.20), **SGU** (see 6.6.21) are user defined values of maximal water (and gas) saturation in current block,

- $\tilde{S}_W$, $\tilde{S}_G$ are scaled block water and gas saturations.

1. Water function rescaling

    Denote

    - in 3phase systems $SR = 1 - SOWCR - SGL$, $S_r = 1 - S_{OWcr} - S_{G\,co}$
    - in oil-water systems $SR = 1 - SOWCR$, $S_r = 1 - S_{OWcr}$

    Then water saturation is rescaled according to formula

    $$\tilde{S}_W = \begin{cases} S_{Wcr} + \frac{(S_W - SWCR)(S_r - S_{Wcr})}{SR - SWCR} & SWCR < S_W < SR \\ S_r + \frac{(S_W - SR)(S_{Wmax} - S_r)}{SWU - SR} & SR < S_W < SWU \end{cases} \tag{2.28}$$

    Relative permeability is calculated as in (2.20).

2. Gas function rescaling

    Denote

    - in 3phase systems $SR = 1 - SOGCR - SWL$, $S_r = 1 - S_{ogcr} - S_{W\,co}$
    - in gas-water systems $SR = 1 - SWCR$, $S_r = 1 - S_{Wcr}$

    Then gas saturation is rescaled according to formula

    $$\tilde{S}_G = \begin{cases} S_{Gcr} + \frac{(S_G - SGCR)(S_r - S_{Gcr})}{SR - SGCR} & SGCR < S_G < SR \\ S_r + \frac{(S_G - SR)(S_{Gmax} - S_r)}{SGU - SR} & SR < S_G < SGU \end{cases} \tag{2.29}$$

    Relative permeability is calculated as in (2.21).

The keyword **TZONE** (see 6.6.12) controls the transition zone option. If the parameter is set – true to a phase, then the critical saturations for that phase will be modified to be the initial immobile saturation in regions where the saturation is below the input critical value.

1. the parameter is set true to oil phase – **SOWCR** (see 6.6.18) will be modified for oil-water runs or oil-water-miscible gas runs, **SOGCR** (see 6.6.19) will be modified only for oil-gas runs;

2. the parameter is set true to water phase, **SWCR** (see 6.6.16) will be modified;

3. the parameter is set true to gas phase, **SGCR** (see 6.6.17) will be modified for gas-water runs and oil-gas runs.

Relative permeabilities scaling
If at least one of the following arrays **KRO** (see 6.6.25), **KRW** (see 6.6.26), **KRG** (see 6.6.27), **KRORW** (see 6.6.25), **KRORG** (see 6.6.25), **KRWR** (see 6.6.26), **KRGR** (see 6.6.27) is set, relative permeabilities are additionaly scaled according to formulas, the same as for two-point scaling case, see (2.22) - (2.27). The only difference is that now SR and SPCR are defined properly.

2.7 Equation of state

In case of compositional run oil and gas properties are calculated from equation of state (**EOS** (see 6.7.5)):

$$Z^3 + E_2 Z^2 + E_1 Z + E_0 = 0 \qquad (2.30)$$

Maximal positive root of equation (2.30) is equal to vapor Z-factor (correspondingly, minimal positive root for liquid phase Z-factor).

Equation (2.30) coefficients are calculated as follows.

For every component (**CNAMES** (see 6.7.4)) user defines

- T_{ci} - component i critical temperature, **TCRIT** (see 6.7.11),

- p_{ci} - component i critical pressure, **PCRIT** (see 6.7.12),

- ω_i - component i acentric factor, **ACF** (see 6.7.15),

- c_{ij} - binary interaction coefficients, **BIC** (see 6.7.16).

Next, coefficients m_1, m_2, Ω_{a0}, Ω_b are taken from table according to equation of state type:

EOS	m_1	m_2	Ω_{a0}	Ω_b
RK	0	1	0.4274802	0.08664035
SRK	0	1	0.4274802	0.08664035
PR	$1+\sqrt{2}$	$1-\sqrt{2}$	0.457235529	0.07796074

Then basing on current temperature and pressure p, T reduced values are calculated for each component:

$$p_{ri} = p/p_{ci}, \quad T_{ri} = T/T_{ci}. \qquad (2.31)$$

Depending on EOS type, Ω_a is taken as:

- RK

$$\Omega_a(T,i) = \Omega_{a0}T_{ri}^{-0.5} \qquad (2.32)$$

- SRK

$$\Omega_a(T,i) = \Omega_{a0}\left(1 + (0.48 + 1.574\omega_i - 0.176\omega_i^2)(1 - T_{ri}^{0.5})\right)^2 \qquad (2.33)$$

- PR

$$\Omega_a(T,i) = \Omega_{a0}\left(1 + (0.37464 + 1.54226\omega_i - 0.26992\omega_i^2)(1 - T_{ri}^{0.5})\right)^2 \qquad (2.34)$$

Next, the simulator calculates

$$A_i = \Omega_a(T,i)\frac{p_{ri}}{T_{ri}^2}, \quad B_i = \Omega_b\frac{p_{ri}}{T_{ri}}, \quad A_{jk} = (1 - c_{jk})(A_jA_k)^{0.5}. \qquad (2.35)$$

Now EOS coefficients can be calculated:

$$A = \sum_{j=1}^{N}\sum_{k=1}^{n} y_jy_kA_{jk}, \quad B = \sum_{j=1}^{N} y_jB_j \qquad (2.36)$$

$$\begin{aligned}
E_2 &= (m_1 + m_2 - 1)B - 1, \\
E_1 &= A - (2(m_1 + m_2) - 1)B^2 - (m_1 + m_2)B, \\
E_0 &= -\left(AB + m_1m_2B^2(B+1)\right)
\end{aligned} \qquad (2.37)$$

2.8 Phase viscosity

Black oil: Phase viscosity $\mu_P = \mu_P(p_P)$ is user specified function. For oil and gas phases it is specified in a number of points p_P and is interpolated for other points (PVDO (see 6.5.1), PVTO (see 6.5.3), PVCDO (see 6.5.2), PVCO (see 6.5.5) for oil phase and PVDG (see 6.5.6), PVTG (see 6.5.7), PVZG (see 6.5.8) for gas phase). For water phase it is specified in a single point accompanied by pressure derivative (PVTW (see 6.5.4)). In case of absence of laboratory measurements the approximations obtained from PVT properties by correlation analysis may be used.

Compositional: For water phase viscosity is constant (PVTW (see 6.5.4)). For hydrocarbon phases we use Lohrenz-Bray-Clark correlation

$$\left((\mu_P - \mu_P^*)\chi + 10^{-4}\right)^{1/4} = a_1 + a_2\xi_{rP} + a_3\xi_{rP}^2 + a_4\xi_{rP}^3 + a_5\xi_{rP}^4 \qquad (2.38)$$

Here $\xi_{rP} = \xi_P/\xi_c$, coefficients a_i are equal to

$$\begin{aligned}
a_1 &= 0.1023000, \quad a_2 = 0.0233640, \quad a_3 = 0.0585330, \\
a_4 &= -0.0407580, \quad a_5 = 0.0093324.
\end{aligned} \qquad (2.39)$$

and

$$\chi = \left(\sum_{i=1}^{N} z_i T_{ci}\right)^{1/6} \left(\sum_{i=1}^{N} z_i M_{wi}\right)^{-1/2} \left(\sum_{i=1}^{N} z_i p_{ci}\right)^{-2/3}. \quad (2.40)$$

Critical temperatures T_{ci} (TCRIT (see 6.7.11)), critical pressures p_{ci} (PCRIT (see 6.7.12)) and molecular weights M_{wi} (MW (see 6.7.14)) are user defined.

Phase molar density is ξ_P is defined in 2.10. Critical density ξ_c could be found from user entered critical volumes V_{ci}, VCRIT (see 6.7.13):

$$\xi_c = \left(\sum_{i=1}^{N} z_i V_{ci}\right)^{-1}. \quad (2.41)$$

Dilute gas mixture viscosity μ^* is calculated from

$$\mu^* = \left(\sum_{i=1}^{N} z_i \mu_i^* M_{wi}^{1/2}\right) \left(\sum_{i=1}^{N} z_i M_{wi}^{1/2}\right)^{-1}. \quad (2.42)$$

where dilute gas viscosities for individual component i, μ_i^*, are defined as

$$\mu_i^* = \begin{cases} 34 \times 10^{-5} T_{ri}^{0.94}/\chi_i, & T_{ri} \le 1.5 \\ 17.78 \times 10^{-5} (4.58 T_{ri} - 1.67)^{0.625}/\chi_i, & T_{ri} > 1.5 \end{cases} \quad (2.43)$$

and

$$\chi_i = T_{ci}^{1/6} M_{wi}^{-1/2} p_{ci}^{-2/3}. \quad (2.44)$$

2.9 Phase formation volume factor

Phase formation volume factor $B_P = B_P(p_P)$ is user specified function. For oil and gas phases it is specified in a number of points p_P and is interpolated for other points (PVDO (see 6.5.1),PVTO (see 6.5.3), PVCDO (see 6.5.2), PVCO (see 6.5.5) for oil phase and PVDG (see 6.5.6), PVTG (see 6.5.7), PVZG (see 6.5.8) for gas phase). For water phase it is a single point accompanied by pressure derivative (compressibility) (PVTW (see 6.5.4)). In case of absence of laboratory measurements the approximations obtained from PVT properties by correlation analysis may be used.

2.10 Phase molar density

Black oil: Phase molar density is always function of B_P - formation volume factor. For water component it is calculated as

$$\xi_W = \frac{\xi_{W,SC}}{B_W} \quad (2.45)$$

Here $B_W = B_W(p_W)$ – formation volume factor for water, user specified function. For water user specifies formation volume factor at reference pressure and compressibility that is defined as:

$$c_W = -\frac{1}{B_W}\frac{\partial B_W}{\partial p} \tag{2.46}$$

For oil and gas phases molar density in black oil assumptions is calculated as

$$\xi_O = \frac{R_{G,O}\xi_{G,SC} + \xi_{O,SC}}{B_O}, \quad \xi_G = \frac{R_{O,G}\xi_{O,SC} + \xi_{G,SC}}{B_G} \tag{2.47}$$

Here $\xi_{P,SC}$ is molar density of phase P in standard conditions.

Compositional: Water molar density is calculated as in black oil case. For RK, SRK and non-shifted PR EOS (EOS (see 6.7.5)) phase molar density is found from equation

$$\xi_P = \frac{p}{ZRT} \tag{2.48}$$

Critical Z-factor is taken from solution of (2.30) for corresponding phase.

In case of shifted PR EOS formula (2.49)

$$\xi_P = 1 \bigg/ \left(\frac{ZRT}{p} - \sum_{i=1}^{n} x_{iP}b_i s_i\right) \tag{2.49}$$

is used for liquid phase. Here s_i are shift parameters, b_i are taken from

$$b_i = \Omega_b \frac{RT_{ci}}{p_{ci}}, \tag{2.50}$$

(see section 2.7 for Ω_b definition), and x_{iP} are molar fractions of component i in phase P.

2.11 Phase mass density

Black oil: Phase mass density ρ_P is obtained by user input data. Usually user specifies $\rho_{P,SC}$ – phase mass density at surface condition. In case of absence of laboratory measurements the approximations obtained from PVT properties by correlation analysis may be used.

Very often the following constants are known (user input, DENSITY (see 6.5.15)):

- $\rho_{O,SC}$ – oil density at surface condition
- $\rho_{G,SC}$ – gas density at surface condition
- $\rho_{W,SC}$ – water density at surface condition

Then the following functions by analogy with phase molar density are used as $\rho_P(p_P)$ in black oil model:

$$\rho_W = \frac{\rho_{W,SC}}{B_W} \tag{2.51}$$

$$\rho_O = \frac{R_{G,O}\rho_{G,SC} + \rho_{O,SC}}{B_O} \tag{2.52}$$

$$\rho_G = \frac{R_{O,G}\rho_{O,SC} + \rho_{G,SC}}{B_G} \tag{2.53}$$

Compositional: Mass density of water is calculated as in black oil case. Mass density of hydrocarbon component is calculated from molar density (see 2.10) as

$$\rho_P = \xi_P M_{wP}, \tag{2.54}$$

where M_{wP} is average molecular weight for phase P,

$$M_{wO} = \sum_{i=1}^{N} M_{wi}x_{iO}, \quad M_{wG} = \sum_{i=1}^{N} M_{wi}x_{iG} \tag{2.55}$$

M_{wi} are component molecular weights, MW (see 6.7.14), and x_{iP} are concentrations of component i in phase P.

2.12 Capillary pressures

2.12.1 Oil-gas capillary pressure

Oil-gas capillary pressure $P_{cOG} = P_{cOG}(S_G)$ is user input data array defined in a number of S_G points and interpolated for other points (SGOF (see 6.6.2)).

2.12.2 Oil-water capillary pressure

Oil-water capillary pressure $P_{cOW} = P_{cOW}(S_W)$ is user input data array defined in a number of S_W points and interpolated for other points (SWOF (see 6.6.1)).

2.12.3 Capillary pressure end-point scaling

If end-point scaling is selected by ENDSCALE (see 6.6.11), capillary pressures are scaled with accordance to user defined arrays of minimal and maximal saturations. Capillary pressures are scaled by two-point method, saturations are renormalilzed according to formulas below.

Let's first introduce notations

- S_W, S_G are block water and gas saturations,

- $S_{W\ co}$, $S_{G\ co}$ are connate water and gas saturations, i.e. minimal water (gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)),

- S_{Wmax}, S_{Gmax} are maximal values of water (and gas) saturation in SWOF (see 6.6.1) (SGOF (see 6.6.2)),

- SWL (see 6.6.14), SGL (see 6.6.15) are user defined values of connate water (and gas) saturation in current block,

- SWU (see 6.6.20), SGU (see 6.6.21) are user defined values of maximal water (and gas) saturation in current block,

- PCW (see 6.6.28), PCG (see 6.6.29) are user defined arrays of maximum capillary pressures,

- $P_{cOW\ max}(table)$, $P_{cOG\ max}(table)$ are maximum capillary pressures from table SWOF (see 6.6.1) (SGOF (see 6.6.2)) (values at connate water and gas saturations)

- $\tilde{S}_W$, $\tilde{S}_G$ are scaled block water and gas saturations.

Saturations are scaled as

$$\tilde{S}_W = S_{W\ co} + \frac{(S_W - SWL)(S_{Wmax} - S_{W\ co})}{SWU - SWL} \tag{2.56}$$

$$\tilde{S}_G = S_{G\ co} + \frac{(S_G - SGL)(S_{Gmax} - S_{G\ co})}{SGU - SGL}$$

Appropriate capillary pressures are caculated as

$$P_{cOW}(S_W) = P_{cOW}(\tilde{S}_W)(table) * \frac{PCW}{P_{cOW\ max}(table)} \tag{2.57}$$

$$P_{cOG}(S_G) = P_{cOG}(\tilde{S}_G)(table) * \frac{PCG}{P_{cOG\ max}(table)} \tag{2.58}$$

2.12.4 Capillary pressure calculation according to Leverett J-function

If end-point scaling is selected by ENDSCALE (see 6.6.11), capillary pressures may be calculated according to Leverett J-function model. Phase capillary pressures are scaled if first argument of JFUNC (see 6.2.47) indicates scaling is to be performed. Scaling formulas are the following:

$$P_{cOW}(S_W) = J_W(S_W)(table) * J_{mult\ W} \tag{2.59}$$

$$P_{cOG}(S_G) = J_G(S_G)(table) * J_{mult\ G} \tag{2.60}$$

Here J_W, J_G are input in the fourth column of SWOF (see 6.6.1) and SGOF (see 6.6.2) keywords, as functions of saturations, in place of phase capillary pressures. Multipliers are calculated as

$$J_{mult\ W} = ST_W * (\phi^{i,j,k})^{\alpha} / (K^{i,j,k})^{\beta} * 0.318316 \tag{2.61}$$

$$J_{mult\ G} = ST_G * (\phi^{i,j,k})^{\alpha} / (K^{i,j,k})^{\beta} * 0.318316$$

where

- ST_W oil-water surface tension, second argument of keyword JFUNC (see 6.2.47);

- ST_G oil-gas surface tension, third argument of keyword JFUNC (see 6.2.47);

- $\phi^{i,j,k}$ porosity in grid block;

- $K^{i,j,k}$ permeability, calculated according to one of the following methods

    - XY: $K^{i,j,k} = (k_{xx}^{i,j,k} + k_{yy}^{i,j,k})/2$
    - X: $K^{i,j,k} = k_{xx}^{i,j,k}$
    - Y: $K^{i,j,k} = k_{yy}^{i,j,k}$
    - Z: $K^{i,j,k} = k_{zz}^{i,j,k}$;

    the method is selected by sixth argument of keyword JFUNC (see 6.2.47);

- α power for porosity, fourth argument of keyword JFUNC (see 6.2.47);

- α power for permeability, fifth argument of keyword JFUNC (see 6.2.47).

2.13 Solubility of gas component into oil phase

Solubility of gas component into oil phase $R_{G,O} = R_{G,O}(p_O)$ (gas-oil ratio) is user input data array defined in a number of p_O points and interpolated for other points (PVTO (see 6.5.3), PVCO (see 6.5.5)).

The keyword DRSDT (see 6.10.29) sets maximum rate of increase of solution gas-oil ratio ($sm^3/sm^3/day$).

2.14 Vaporisation of oil component into gas phase

Vaporisation of oil component in gas phase $R_{O,G} = R_{O,G}(p_G)$ is user input data array defined in a number of p_G points and interpolated for other points (PVTG (see 6.5.7)).

2.15 Well

2.15.1 Well approximation

Well is approximated differently depending on computation mesh. Let us consider source of phase $Q_P = Q_P(p_P, N, t)$ in block l in the case of uniform computation mesh and finite difference approximation. We define Q_P on surface of cylinder of radius r_w with perforated well region as its axis as

$$Q_P(p_P, N, t) = T(t) M_P(p_P, S_W, S_G)(p_P - p_{BH}(t) - \bar{\rho}_{av}(p, N) g(D - D_{BH})) \qquad (2.62)$$

where

- $M_P(p_P, S_W, S_G)$ – phase mobility, **known**, will be defined below, see section 5.3.5,

- $p_{BH}(t)$ – bottom hole pressure, **known** or **calculated** from the value $q(t)$ of user defined well rate,

- $\bar{\rho}_{av}(p, N)$ – average wellbore density, depends on discrete approximation chosen for the equations (2.1)–(2.4), and will be defined below, see section 5.3.6 (**known**)

- D, $g = \text{const}$ have been defined before,

- D_{BH} – bottom hole depth (**known**)

- $T(t)$ – well productivity index (**known**), may be defined by user (COMPDAT (see 6.10.6)), otherwise calculated according to $T = \dfrac{2\pi K_{mult}(t)\beta_c Kh}{(\log(r_0/r_w) + s)}$, see section 5.3.2. Here

  - $K_{mult}(t)$ – multiplier (**known**, WPIMULT (see 6.10.10))

  - $\beta_c = \text{const}$ – units conversion factor (**known**)

  - Kh (**known**) – may be defined by user (COMPDAT (see 6.10.6)) or calculated as product of $h = \text{const}$, well perforated interval height (**known**), and K – permeability in plane perpendicular to well axis, depends on the discrete approximation chosen for the equations (2.1)–(2.4), and will be defined below, see section 5.3.3 (**known**)

  - r_0 is pressure equivalent radius, may be defined by user, COMPDAT (see 6.10.6), otherwise its approximation depends on the discrete approximation chosen for the equations (2.1)–(2.4), and will be defined below, see section 5.3.4.

  - $r_w = d_w/2 = \text{const}$ – well radius (**known**, COMPDAT (see 6.10.6))

  - $s = s(x, y, z, t)$ – skin effect term (**known**, COMPDAT (see 6.10.6))

In this case source of component c will be equal to

$$q_c = \sum_P x_{c,P} \xi_P Q_P(p, N) \tag{2.63}$$

where

- $x_{c,P} = x_{c,P}(p, N)$ – moles of component c per mole of phase P,

- $\xi_P = \xi_P(p, N)$ – phase molar density, see 2.10,

- $Q_P(p, N)$ – phase rate in reservoir conditions, calculated in (2.62).

The keyword WRFT (see 6.10.66) sets output of well RFT data. The following data will be written to RFT file: pressure, saturation and depth for each grid block in which a well has a connection.

2.15.2 Multisegment well

The structure of multisegment well is specified via the keyword WELSEGS (see 6.10.4). WSEGDIMS (see 6.1.22) – dimensions of data for multisegment well (not necessarily keyword, tNavigator allocates memory dynamically). WSEGTABL (see 6.10.5) – specifies calculation of segment pressure drops from VFP tables. Definition of location of completions in a multisegment well – COMPSEGS.

2.16 NETWORK option. Automatic chokes. Compressors

NETWORK option is specified via the keyword NETWORK (see 6.1.42). The keyword sets dimensions for extended network model. The extended network model is specified via keywords NODEPROP (see 6.10.27), BRANPROP (see 6.10.26).

NETWORK option is used to provide variable THP limits to groups of wells, which depend on the groups' flow rates according to a set of pipeline pressure loss relationships. If NETWORK option is used well THP limits will be calculates dynamically by balancing the flow rates and pressure losses in the network.

Tree-structure for groups should be defined.

GRUPTREE (see 6.10.25) – sets tree structure for multi-level group control. The tree can consist of an arbitrary number of levels. The field FIELD occupies the top of this tree. Groups that have other groups as children cannot have wells. (Wells are assigned to groups in the keyword WELSPECS (see 6.10.2)). Thus a group either contains wells (that is a wellgroup) or has other groups as children (that is a node-group).

In case of extended network model group structure can be different from the structure specified by GRUPTREE (see 6.10.25) (the bottom nodes in the tree should be the same (i.e. well groups)). BRANPROP (see 6.10.26) specifies branch properties. NODEPROP (see 6.10.27) specifies node properties. The top node should have a fixed pressure.

The network can consist of two or more separate trees. Each tree should have its own fixed pressure terminal node at the top. Child groups also can be nodes with fixed pressure (so pressures in sub-networks are independent of the main network but flows of sub-networks will be added into main network flow).

Automatic chokes.

An automatic choke can adjust the pressure loss across a choke in a designated network branch to meet a group's production rate target.

A branch is set as a choke via a flag YES in parameter 3 of the keyword NODEPROP (see 6.10.27) for the inlet node of the choke. The branch should have a number 9999 as a

corresponding VFP table number (in the keyword BRANPROP (see 6.10.26)).

Automatic compressors (pumps).

Compressors are specified via the keyword NETCOMPA (see 6.10.28). Compressors (pumps) are turned on if a nominated group cannot meet its production rate target (which is specified with keyword GCONPROD (see 6.10.23)). Multi-level compressors can be specified (compression is increased one level at a time until the nominated group can meet its production target).

2.17 Polymer flooding based on BrightWater technology

Flow deflecting technologies – is a way to increase efficiency of reservoir development. For example these technologies can base on nanopolymer flooding. BrightWater technology (developed by companies BP, Chevron and Nalco) consists in nanopolymer injection into the formation. Particles of nanopolymers increase theirs volume (average – in 10 times) at hydrolysis or heating. The main idea is that small granules (average size – 100 nm) are injected into the formation with water phase.

Granule size is considerably smaller than pore size at 500 mD (or more) permeability. Particles of polymers expand in formation pores in the direction from producers to injectors. Pores in zones of active filtration are blocked and water is forced out to the zones with low permeability. This process is called activation of nanopolymer.

In tNavigator there is nanopolymer flooding option. The following physical effects are taken into consideration:

- hydrolysis swelling rate,

- nanopolymer type.

Simulator uses standart isothermic black-oil model (2.1)–(2.4), nanopolymer is considered as an admixture to water phase, that changes filtration-capacity formation properties. Let C_{pol} and t_{pol} – nanopolymer concentration in water phase and nanopolymer holding time in the formation. Within the bounds of this BrightWater model we consider that absolute permeability is: $k = k_{mult}(C_{pol}, t_{pol})k_0$, where k_0 — initial absolute permeability (that was in the model before nanopolymer flooding), $k_{mult}(C_{pol}, t_{pol})$ can be presented the following way:

$$k_{mult}(C_{pol}, t_{pol}) = 1 - (1 - k_{conc}(C_{pol}))(1 - k_{time}(t_{pol})),$$

where $k_{conc}(C_{pol})$ and $k_{time}(t_{pol})$ are defined via tables TRMMULTC (see 6.5.28), TRMMULTT (see 6.5.29).

2.18 Inflow from aquifer

tNavigator 3.2 supports the following type of aquifers:

- numerical aquifer. Is set via the keywords AQUCON (see 6.11.9), AQUNUM (see 6.11.8);

- constant-flux aquifer (analytic aquifer). Is set via the keywords AQUFLUX (see 6.11.2), AQUANCON (see 6.11.7);

- Fetkovich aquifer (analytic aquifer). Is set via the keywords AQUFETP (see 6.11.4), AQUANCON (see 6.11.7), AQUFET (see 6.11.3);

- Carter-Tracy aquifer (analytic aquifer). Is set via the keywords AQUTAB (see 6.11.6), AQUANCON (see 6.11.7), AQUCT (see 6.11.5).

Brine option is supported for aquifers (BRINE (see 6.1.33)) (salt concentration is set via keywords AQUFETP (see 6.11.4), AQUFET (see 6.11.3), AQUCT (see 6.11.5)).

2.19   Sweet water injection into the saline reservoir

Salination of reservoir production layers – is the way to localize residual reserves of hydrocarbons, determine the efficiency of reservoir development, determine oil
Flooding efficiency depends on pore filling of salt and depends on salt solubility in the sweet water.
tNavigator has an option – simulation of sweet water injection into the saline reservoir. Option BRINE (see 6.1.33) – This keyword indicates that the Brine Traking option is enable, to allow the modeling of waters with different salinities.
Salt washing-out with sweet water is simulated the following way:

- ROCKSALT (see 6.9.27) (mass of reservoir salt that can be dissolved kg);

- spreading of injected water is calculated (water salinity is specified via WSALT (see 6.10.62));

- reservoir salt is dissolved that leads to formation porosity changes and to increasing of salt concentration in the water, water density and viscosity increase too (see PVTWSALT (see 6.5.30));

- oil viscosity and water viscosity are equalized that leads to improvement of oil forcing out;

- ultimate concentration of dissolved (in the water) salt – SALTPROP (see 6.5.26);

- reservoir salt dissolution rate is directly proportional to difference of salt solution (current and saturated).

The keyword SALTTRM (see 6.5.27) sets the dependence between permeability and amount of dissolved reservoir salt. SALTTRM (see 6.5.27) and SALTPROP (see 6.5.26) can be specified for different PVT regions.

The initial salt concentration (kg/m^3) can be specify via the keyword SALT (see 6.9.28). The keyword should be used when the initial state has been set by enumeration (keywords PRESSURE (see 6.9.7), RS (see 6.9.19), RV (see 6.9.20), SGAS (see 6.9.10) and SWAT (see 6.9.9)). For a run initialized by equilibration EQUIL (see 6.9.1), the keyword SALTVD (see 6.9.29) should be used instead of SALT (see 6.9.28) (salt concentration versus depth for equilibration).

Brine option is supported for aquifers (BRINE (see 6.1.33)) (salt concentration is set via keywords AQUFETP (see 6.11.4), AQUFET (see 6.11.3), AQUCT (see 6.11.5)).

2.20 Dual porosity

Dual porosity is specifying by the keyword DUALPORO (see 6.1.37).
Dual permeability — DUALPERM (see 6.1.38).

Dual porosity. In a resevoir with the dual porosity there are two systems: rock matrix (the biggest part of the reservoir) and fractures (which have high permeability).

Dual porosity single permeability: fluid flow between matrix cells is possible only using fractures. Fluid flow through the reservoir is possible only in fractures.
Dual porosity dual permeability: fluid flow between neighboring matrix cells is possible.

If these options are used, for every geometric grid block we consider two cells: the matrix part and the fracture part of this block. One can specify their properties (porosity, permeability etc.) independently.

If the keyword DUALPORO (see 6.1.37) or DUALPERM (see 6.1.38) is used, the number of layers in the Z-direction should be even (this number is entered by the third parameter of the keyword DIMENS (see 6.1.12) (NZ). The first half of the grid blocks corresponds to the matrix cells, and the second half – fracture cells. tNavigator autimatically create non-neighbor connections which correspond to the matrix-fracture flows.

The keyword NODPPM (see 6.1.41) cancels a multiplication of permeability (for the fracture blocks) by porosity (fracture blocks) during the dual porosity run. Since this multiplication is used to obtain a net bulk fracture permeability one have to enter this value manually if NODPPM is enable.

DPNUM (see 6.2.51) – specifies reservoir fields that should be considered as single porosity fields.

DPGRID (see 6.2.52) – if the keyword is enable one should specify grid data only for matrix blocks (NX * NY * (NZ/2)); values for fracture blocks will be obtained (copied) from corresponding matrix blocks.
This operation is applied for the values specified by following keywords: DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1), PERMX (see 6.2.7), PERMY (see 6.2.7), PERMZ (see 6.2.7),

PORO (see 6.2.17), TOPS (see 6.2.5), NTG (see 6.2.18), DZNET (see 6.2.19), ZCORN (see 6.2.33), PERMXY / PERMYZ / PERMXZ (see 6.2.8), DEPTH (see 6.2.6). This operation is applied only for fracture blocks which don't have manually input grid data.

Transmissibility calculations in dual porosity runs. Matrix-fracture transmissibility is calculated via the formula:

$$Tr = CDARCY * K * V * \sigma$$

where K – X-direction permeability of the matrix blocks X,
V – matrix cell bulk volume,
σ – sigma-factor.

σ sigma-factor can be specified for whole reservoir – SIGMA (see 6.2.53) or different values for different grid blocks can be entered – SIGMAV (see 6.2.54).

Sigma-factor is related to the distances between fractures (matrix block sizes) in X, Y and Z directions:

$$\sigma = 4(\frac{1}{lx^2} + \frac{1}{ly^2} + \frac{1}{lz^2}),$$

lx, ly и lz – the distances between fractures (matrix block sizes) in X, Y and Z directions. (These distances are not the dimensions DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1).)

Default. If no one of the keywords SIGMA, SIGMAV (see 6.2.54), LTOSIGMA (see 6.2.55) is specified, sigma-factor will be considered as zero.

Viscous Displacement.
VISCD (see 6.1.40) – the keyword sets that the Viscous displacement option will be used in the dual porosity run.

If this option is used, one should specify the distances between fractures (matrix block sizes) in X, Y and Z directions using keywords LX (see 6.2.50), LY (see 6.2.50), LZ (see 6.2.50).

Viscous displacement – fluid flow under the influence of pressure gradient. One can observe a pressure gradient in the dual porosity system. This gradient moves the fluid in the fracture towards the production well. If this gradient is small and fracture permeability is high, the matrix-fracture viscous displacement under the influence of pressure gradient isn't considered. Nevertheless, if fractions have small permeability then the matrix-fracture viscous displacement under the influence of pressure gradient can be very important in production.

tNavigator can compute a σ factor using keywords LX (see 6.2.50), LY (see 6.2.50), LZ (see 6.2.50) and LTOSIGMA (see 6.2.55). If tNavigator compute σ, any manually input of this parameter (SIGMA (see 6.2.53)) in data file will be ignored.

LTOSIGMA (see 6.2.55) – this keyword can be used in dual porosity run, if the option Viscous displacement is enable (VISCD (see 6.1.40)). Using the keyword LTOSIGMA (see 6.2.55) sigma-factor multiplier can be obtained from the distances between fractures (matrix block sizes).

Sigma-factor is related to the distances between fractures (matrix block sizes) in X, Y and Z directions:

$$\sigma = \frac{fx}{lx^2} + \frac{fy}{ly^2} + \frac{fz}{lz^2},$$

lx, ly и lz – the distances between fractures (matrix block sizes) in X, Y and Z directions. (These distances are not the dimensions DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1).) The values of lx, ly, lz that aren't specified or are equal to zero will not be used in calculations. LTOSIGMA (see 6.2.55) defines fx, fy, fz.

Multipliers for sigma-factor.

Sigma-factor (defined via SIGMA (see 6.2.53), SIGMAV (see 6.2.54)) is multiplied by the multiplier MULTSIG (see 6.10.37) (the same multiplier for whole reservoir) or different multipliers for grid blocks can be entered using the keyword MULTSIGV (see 6.10.38).

MULTMF (see 6.2.56) – the keyword specifies multiplier which is used to calculate the matrix-fracture flows.

2.21 LGR – Local Grid Refinement

In tNavigator local grid refinements can be specified.

The following keywords are supported:

- LGR (see 6.1.39) (RUNSPEC section) – set options and dimensions for local grid refinement.

- CARFIN (see 6.2.61) (GRID section)– specifies a cartesian local grid refinement (LGR). CARFIN (see 6.2.61) specifies a cell or a box of cells identified by its global grid coordinates I1-I2, J1-J2, K1-K2, to be replaced by refined cells. CARFIN (see 6.2.61) can be followed by keywords that describe properties in LGR, if they are different from the properties in parent grid. These keywords should be terminated with the keyword ENDFIN (see 6.2.63), which terminates data for a local grid refinement.

- REFINE (see 6.2.62) – initiates data input for a named local grid (LGR). The keyword should be followed by name of local grid refinement the data for which is entered. The data should be terminated with the keyword ENDFIN (see 6.2.63), which terminates data for a local grid refinement. REFINE can be used in GRID, EDIT, PROPS, REGIONS, SOLUTION and SCHEDULE section.

- ENDFIN (see 6.2.63) – terminates data for a local grid refinement.

- NXFIN / NYFIN / NZFIN (see 6.2.64) (GRID section) – These keywords can be used to specify number of local cells in each global cell of an LGR (NXFIN – in X direction, NYFIN – in Y direction, NZFIN – in Z direction).

- HXFIN / HYFIN / HZFIN (see 6.2.65) (GRID section) – These keywords can be used to specify the size ratios of each cell in a local grid refinement (LGR) (HXFIN – in X direction, HYFIN – in Y direction, HZFIN – in Z direction).

- WELSPECL (see 6.10.3) (SCHEDULE section) – introduces a new well, defining information on its name and coordinates in local grids (LGR). WELSPECL must be used in place of WELSPECS (see 6.10.2) to set the general specification data for wells in local refined grids.

- COMPDATL (see 6.10.7) (SCHEDULE section) – defines well completions in local grids (LGR).
  COMPDATL (see 6.10.7) must be used in place of COMPDAT (see 6.10.6) to specify the connection data for wells in local refined grids.

- COMPLMPL (see 6.10.9) (SCHEDULE section) – lumps connections together into completions to provide realization of simultaneous actions for wells in local grids (LGR).
  COMPLMPL (see 6.10.9) must be used in place of COMPLUMP (see 6.10.8) to lump connections together into completions to provide realization of simultaneous actions for wells in local refined grids.

- WPIMULTL (see 6.10.11) (SCHEDULE section) – multiplies well connection transmissibility factors by specified value for wells in local grids (LGR).
  WPIMULTL (see 6.10.11) must be used in place of WPIMULT (see 6.10.10) to multiply well connection transmissibility factors by specified value for wells in local refined grids.

- WFRACL (see 6.10.42) (SCHEDULE section) – specifies the hydraulic fracture for wells in local refined grids (LGR). WFRACL (see 6.10.42) must be used in place of WFRAC (see 6.10.41) to specify the hydraulic fracture for wells in local refined grids.

- WFRACPL (see 6.10.44) (SCHEDULE section) – specifies the hydraulic fracture for wells in local refined grids (LGR) in graphical interface. WFRACPL (see 6.10.44) must be used in place of WFRACP (see 6.10.43) to specify the hydraulic fracture for wells in local refined grids in graphical interface.

- COMPFRACL (see 6.10.46) (SCHEDULE section) – specifies the hydraulic fracture for connection in the grid layer for wells in local refined grids (LGR). COMPFRACL (see 6.10.46) must be used in place of COMPFRAC (see 6.10.45) to specify the hydraulic fracture for connection in the grid layer for wells in local refined grids.

3   Compositional thermal model with chemical reactions

Let's consider a compositional thermal model with chemical reactions which is used in tNavigator. There are threee (four) phases and three (four) or more components:

- water phase (**water**)— doesn't mix with hydrocarbon phases, consists of one component – **water**;

- liquid hydrocarbon phase (**oil**) — consists of a mixture of **hydrocarbon components**, at certain pressure, temperature and with a certain concentration of components in the liquid phase;

- gas hydrocarbon phase (**gas**) — consists of a mixture of **hydrocarbon components**, **oxygen-component**, **water-component**, at certain pressure, temperature and with a certain concentration of components in the gas phase.

- **solid** phase (**coke**) – consists of one component – **coke**.

The following phase changes are supported:

water	⇒	**gas**	—vaporization
gas	⇒	**water**	—condensation
gas	⇒	**oil**	—solution
oil	⇒	**gas**	—oil evaporation
oil	⇒	**coke**	—carbonization
oil, **gas**	⇒	**gas, water**	—burning
coke, **gas**	⇒	**gas, water**	—burning

The solid phase is specified using the keyword SOLID (see   6.1.32) in e300 data format; MODEL (see   8.4.1) – in stars data format. Component volatility type is specified using CVTYPE (see   6.8.1) in e300; MODEL (see   8.4.1).

3.1   Basic volumes

The unit grid block volume V_b is

$$V_b = V_R + V_p, \quad V_p = V_S + V_f, \quad V_f = V_W + V_O + V_G$$

where

- V_R – **rock volume** (is used in the description of thermal properties),

- V_p – **pore volume**, V_f – **"mobile" volume**,

- V_S – **solid phase volume (solid phase)**,

- V_P, $P = W, O, G$ – water, oil, gas **phase volume**.

Porosity ϕ — the volume which can be filled with a mixture:

$$\phi = \frac{V_p}{V_b} = \frac{V_S + V_f}{V_b} = \frac{V_S + V_f}{V_R + V_S + V_f}$$

"Mobile" porosity ϕ_f — the volume, which can be filled with a mobile mixture:

$$\phi_f = \frac{V_f}{V_b} = \frac{V_f + V_S - V_S}{V_b} = \phi - \frac{V_S}{V_b} = \phi \left(1 - \frac{V_S}{V_f + V_S}\right) = \phi \left(1 - \frac{V_S}{V_p}\right)$$

3.2 Saturations

The saturation S_P of liquid phase $(P = W, O, G)$ — a part of volume of porous medium (which can be filled with liquid phases), which is filled with this phase:

$$S_P = \frac{V_P}{V_f} = \frac{V_P}{V_W + V_O + V_G}, \quad P = W, O, G, \quad S_W + S_O + S_g = 1 \tag{3.1}$$

The saturation $\widehat{S}_S$ **of solid phase** is

$$\widehat{S}_S = \frac{V_S}{V_p}$$

so

$$\phi_f = \phi \left(1 - \widehat{S}_S\right) \tag{3.2}$$

and **modified saturations** $\widehat{S}_P$ of liquid phases $(P = W, O, G)$

$$\widehat{S}_P = \left(1 - \widehat{S}_S\right) S_P, \quad P = W, O, G$$

so

$$\widehat{S}_W + \widehat{S}_O + \widehat{S}_G + \widehat{S}_S = 1$$

3.3 Phases

Each phase P, $P = W, O, G, S$ (Water, Oil, Gas, Solid) has the following parameters (**unknown**, these parameters are calculated during the run):

- $T = T(t, x, y, z)$ – **phase temperature** (all phases are in the thermodynamic equilibrium, therefore all phases have the same temperature at one place);

- $p_P = p_P(t, x, y, z)$ – **pressure** of phase P;

- $S_P = S_P(t, x, y, z)$ $(P = W, O, G)$, $\widehat{S}_S = \widehat{S}_S(t, x, y, z)$ – **saturation** of phase P.

The following equations are used to reduce the number of unknowns:

$$
\begin{aligned}
p_O - p_G &= P_{cOG}, \\
p_O - p_W &= P_{cOW}, \\
S_W + S_O + S_G &= 1.
\end{aligned}
$$

where $P_{cOG} = P_{cOG}(S_g)$ – **capillary pressure** in the system oil-gas, $P_{cOW} = P_{cOW}(S_w)$ – capillary pressure in the system water-oil (known functions).

From here "pressure" is "pressure of oil phase" $p = p_O$. $P_{cP} = -P_{cOP}$, where $P_{cOO} = 0$, $P_{cOS} = 0$ and $p_P = p + P_{cP}$.

3.4 Components

All phases P, $P = W, O, G, S = 1, \ldots, n_P$ can be divided into two groups: "mobile" phases (water, oil, gas) $P = W, O, G = 1, \ldots, n'_P$, $n'_P = n_P - 1$ and solid phase $P = S = n_P$. All components c, $c = 1, \ldots, n_c$ can be divided into two groups:

- $c = 1, \ldots, n'_c$ – the components which can be only in "mobile" phases;

- $c = n'_c + 1, \ldots, n_c$ – the components which can be only in the solid phase.

Let:

- $N_c = N_c(t, x, y, z)$ – **molar density** of the component c, $c = 1, \ldots, n'_c$ in the "mobile" volume (mol/m^3), then $N_c \cdot V_f = N_c \cdot \phi_f \cdot V_b$ – the quantity of the component c in the volume V_b (mol);

- $N_c = N_c(t, x, y, z)$ – **molar density** of the component c, $c = n'_c + 1, \ldots, n_c$ in the pore volume (mol/m^3), then $N_c \cdot V_p = N_c \cdot \phi \cdot V_b$ – the quantity of the component c in the volume V_b (mol).

Component distribution in different phases is set via $n_c \times n_P$ **concentration matrix** $x_{c,P} = x_{c,P}(p_P, N)$, $N = (N_1, \ldots, N_{n_c})$:

$$
\sum_{P=1}^{n'_P} x_{c,P} \xi_P S_P = N_c, \quad c \in \{1, \ldots, n'_c\}, \quad x_{c,S} \xi_S \widehat{S}_S = N_c, \quad c \in \{n'_c + 1, \ldots, n_c\}.
$$

where $\xi_P = \xi_P(p_P, N)$ – **the molar density** of the phase P.

The number of hydcarbon components is specified using the keyword COMPS (see 6.7.3) in e300 data format.

Total number of components and the number of components in water, oil and gas phases are set using the keyword MODEL (see 8.4.1) in stars data format.

Since

$$
\sum_{c=1}^{n_c} x_{c,P} = 1, \quad P \in \{1, \ldots, n_P\}, \qquad x_{c,S} = 0, \quad c \in \{1, \ldots, n'_c\}, \tag{3.3}
$$

$$
x_{c,P} = 0, \quad c \in \{n'_c + 1, \ldots, n_c\}, P \in \{1, \ldots, n'_P\}, \qquad x_{c,W} = 0, \quad c \in \{2, \ldots, n_c\},
$$

hence ($S_S = \widehat{S}_S$):

$$\sum_{P=1}^{n_P} x_{c,P}\xi_P S_P = N_c, \quad c \in \{1,\dots,n_c\},$$

and

$$\widehat{S}_S = \frac{\sum\limits_{c=n'_c+1}^{n_c} N_c}{\xi_S}, \quad x_{c,S} = \frac{N_c}{\sum\limits_{k=n'_c+1}^{n_c} N_k} \quad c \in \{n'_c+1,\dots,n_c\}. \tag{3.4}$$

3.5 Mass and molar water density

In e300 data format **the mass density** and **the molar water density** on default (or if the keyword THANALB (see 6.8.3) is present) are calculated

$$\rho_W = \frac{A_0 + A_1 T + A_2 T^2 + A_3 T^3 + A_4 T^4 + A_5 T^5}{1 + A_6 T} e^{c_{w,p}(p - A_7)}, \quad \xi_W = \frac{1}{MW_W} \cdot \rho_W, \tag{3.5}$$

where

$$\begin{array}{ll}
A_0 = 9998.3952 & A_1 = 169.55176 \\
A_2 = -7.987 \times 10^{-2} & A_3 = -46.170461 \times 10^{-5} \\
A_4 = 105.56302 \times 10^{-8} & A_5 = -280.54353 \times 10^{-11} \\
A_6 = 16.87985 \times 10^{-2} & A_7 = -102
\end{array}$$

If the keyword WATDENT (see 6.8.2) is enable, then

$$\rho_W = \frac{\rho_{w,ref}}{(1 - c_{w,p}(p - p_{w,ref}))(1 + c_{w,1,T}(T - T_{w,ref}) + c_{w,2,T}(T - T_{w,ref})^2)}, \quad \xi_W = \frac{1}{MW_W} \cdot \rho_W \tag{3.6}$$

where

- $\rho_{w,ref} = \rho_{W,SC}/B_W(p_{ref}) \; (kg/m^3)$

- $\rho_{W,SC}$ – is set via DENSITY (see 6.5.15)

- $c_{w,p}$, $p_{w,ref}$, $B_W(p_{ref})$, – is set via PVTW (see 6.5.4)

- $c_{w,1,T}$, $c_{w,2,T}$, $T_{w,ref}$, – is set via WATDENT (see 6.8.2)

- MW_W – water molar weight

In stars data format **the mass density** and **the molar water density** are calculated

$$\xi_W = \rho_{w,ref} \exp\left(c_{w,p}(p - p_{ref}) - c_{w,1,T}(T - T_{ref}) - c_{w,2,T}\frac{T^2 - T_{ref}^2}{2} + c_{w,pT}(p - p_{ref})(T - T_{ref})\right)$$

$$\rho_W = \xi_W \cdot MW_W$$

$$\tag{3.7}$$

where

- p_{ref} – reference pressure PRSR (see 8.4.3)

- T_{ref} – reference temperature TEMR (see 8.4.4)

- $\rho_{w,ref}$ – плотность компоненты w из MOLDEN (see 8.4.5) (mol/m^3)

- $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – the properties of component w specified via CP (see 8.4.7), CT1 (see 8.4.8), CT2 (see 8.4.9), CPT (see 8.4.10).

3.6 Mass and molar liquid density

The molar density and **mass liquid density** are calculated as

$$
\begin{aligned}
\xi_O = 1 \Big/ \left(\sum_{k=2}^{n'_c} x_{k,O} \frac{MW_k}{\rho_{k,O}} \right) \quad & \rho_O = 1 \Big/ \left(\sum_{k=2}^{n'_c} x_{k,O} \frac{1}{\rho_{k,O}} \right) \quad \text{(in e300)} \\
\xi_O = 1 \Big/ \left(\sum_{k=2}^{n'_c} x_{k,O} \frac{1}{\rho_{k,O}} \right) \quad & \rho_O = 1 \Big/ \left(\sum_{k=2}^{n'_c} x_{k,O} \frac{MW_k}{\rho_{k,O}} \right) \quad \text{(in stars)}
\end{aligned}
\tag{3.8}
$$

In stars data format **component liquid density** $\rho_{k,O}(p,T)$ (mol/m^3) is calculated as

$$
\rho_{k,O} = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,1,T}(T - T_{ref}) - c_{k,2,T} \frac{T^2 - T_{ref}^2}{2} + c_{k,pT}(p - p_{ref})(T - T_{ref}) \right)
\tag{3.9}
$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (MOLDEN (see 8.4.5))

4. $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – properties of component k in the liquid phase:

   - $c_{k,p}$ (CP) – component k compressibility k,
   - $c_{k,1,T}$ (CT1) – the first thermal expansion coefficient for component k (for this parameter tNavigator uses this keyword THERMEX1 (see 6.8.25)),
   - $c_{k,2,T}$ (CT2) – the second thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword THERMEX2 (see 6.8.26)); total thermal expansion coefficient is equal to $c_{k,1,T} + T * c_{k,2,T}$,
   - $c_{k,pT}$ (CPT) – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses the keyword THERMEX3 (see 6.8.27))

In e300 data format **component liquid density** $\rho_{k,O}(p,T)$ (kg/m^3) is calculated as

$$
\rho_{k,O} = \rho_{k,ref} \Big/ \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \right)
\tag{3.10}
$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

3.7 Molar and mass gas density

In e300 data format **the molar** and **the gas mass density** are calculated as

$$
\begin{aligned}
\xi_G &= 1 \left/ \left(x_{w,G}\frac{MW_w}{\rho_w} + \sum_{k=2}^{n'_c} x_{k,G}\left(\frac{Z_{k,0}RT}{p} - Z_{k,1}\right) \right) \right. \\
\rho_G &= 1 \left/ \left(\frac{x_{w,G}}{\rho_w} + \sum_{k=2}^{n'_c} \frac{x_{k,G}}{MW_k}\left(\frac{Z_{k,0}RT}{p} - Z_{k,1}\right) \right) \right.
\end{aligned}
\tag{3.11}
$$

The component water density (water vapor) $\rho_w(p,T)$ in the gas phase

$$
\rho_w = \exp\left(\sum_{j=0}^{5} C_j T_{bK}^{j}\right)\frac{T_b}{T}
\tag{3.12}
$$

where T_b – the boiling temperature $°C$, $T_{bK} = T_b + 273.15$ – the boiling temperature $°K$,

$$
\begin{array}{lll}
T_b = a\cdot(p/10)^b, & a = 180.89 & b = 0.2350 \\
C_0 = -93.7072 & C_1 = 0.833941 & C_2 = -0.003208 \\
C_3 = 6.57652\cdot10^{-6} & C_4 = -6.93747\cdot10^{-9} & C_5 = 2.97203\cdot10^{-12}
\end{array}
$$

The coefficients $Z_{k,0}$, $Z_{k,1}$ are specified using the keywords ZFACTOR (see 6.8.32), ZFACT1 (see 6.8.33).

In stars data format **the molar** and **the gas mass density** are calculated

$$
\xi_G = \frac{p}{ZRT}, \quad \rho_G = \xi_G\cdot\sum_{k=1}^{n'_c} x_{k,G}\cdot MW_k
\tag{3.13}
$$

where Z – the root of the equation of state Redlich-Kwong with zero coefficients of pair-wise interaction. Let's specify $i = 1,\ldots,n'_c$

$$
A_i = 0.4274802\frac{p_{ri}}{T_{ri}^{2.5}}, \quad B_i = 0.08664035\frac{p_{ri}}{T_{ri}}, \quad A_{jk} = (A_j A_k)^{0.5}, \quad p_{ri} = \frac{p}{p_{ci}}, \quad T_{ri} = \frac{T}{T_{ci}},
\tag{3.14}
$$

where the critical temperature T_{ci} is set using TCRIT (see 8.4.12), the critical pressure p_{ci} is set using PCRIT (see 8.4.11). Then

$$A = \sum_{j=1}^{n'_c} \sum_{k=1}^{n'_c} x_{j,G} x_{k,G} A_{jk}, \quad B = \sum_{j=1}^{n'_c} x_{j,G} B_j \tag{3.15}$$

Z – the maximal root (> 0) of the equation

$$Z^3 - Z^2 + (A - B^2 - B)Z - AB = 0 \tag{3.16}$$

Z is calculated in every grid block at every time step of Newton iteration. Usually $Z \in [0.3, 1.2]$.

3.8 Molar solid density

Molar solid density (coke) is calculated as

$$\xi_S = 1 \left/ \left(\sum_{k=n'_c+1}^{n_c} x_{k,S} \frac{MW_k}{\rho_k} \right) \right. \quad \text{(in e300)} \quad \text{or} \quad \xi_S = 1 \left/ \left(\sum_{k=n'_c+1}^{n_c} x_{k,S} \frac{1}{\rho_k} \right) \right. \quad \text{(in stars)} \tag{3.17}$$

In e300 data format **component solid density** $\rho_k(p,T)$ is calculated as

$$\rho_k = \rho_{k,ref} \left/ \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \right) \right.$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (is specified using the keyword STHERMX1 (see 6.8.19))

In stars data format **component solid density** $\rho_k(p,T)$ is calculated as

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,T}(T - T_{ref}) + c_{k,pT}(p - p_{ref})(T - T_{ref}) \right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SOLID_DEN (see 8.4.13))

4. $c_{k,p}$, $c_{k,T}$, $c_{k,pT}$ – properties of component k in the solid phase (SOLID_DEN (see 8.4.13)):

   - $c_{k,p}$ – component k compressibility,
   - $c_{k,T}$ – thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword STHERMX1 (see 6.8.19)),
   - $c_{k,pT}$ – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses the keyword STHERMX2 (see 6.8.20))

   Mass solid density isn't used in the calculations.

3.9 Thermodynamic equilibrium condition

M_P - the number of moles in the "mobile" phase P, $P = W, O, G$, in the unit volume. Obviously

$$\sum_{P=1}^{n'_P} M_P = N_{tot} = \sum_{c=1}^{n'_c} N_c.$$

Let $R_P = \dfrac{M_P}{N_{tot}}$ — the part of "mobile" component mixture in the phase P, concerning the total amount of the mixture. Then

$$M_P = R_P \cdot N_{tot}, \quad \sum_{P=1}^{n'_P} R_P = 1. \tag{3.18}$$

Let $M_{c,P} = x_{c,P} \cdot M_P$ — the number of moles of the component c, $c = 1, \ldots, n'_c$, in the phase P, $P = W, O, G$, in the unit volume. Then

$$N_c = \sum_{P=1}^{n'_P} M_P \cdot x_{c,P}, = N_{tot} \sum_{P=1}^{n'_P} R_P \cdot x_{c,P}, \qquad c = 1, \ldots, n'_c.$$

Since water component isn't present in the oil phase and hydrocarbon components are not present in the water phase, then

$$x_{1,O} \equiv x_{w,O} = 0, \qquad x_{c,W} = 0, \quad c \in \{2, \ldots, n'_c\}, \qquad x_{w,W} = 1 \ \left(\text{из} \ \sum_{c=1}^{n'_c} x_{c,P} = 1 \ \text{для} \ P = W\right)$$

Let $z_c = N_c / N_{tot}$, $c = 1, \ldots, n'_c$, then

$$z_w = R_W \cdot x_{w,W} + R_G \cdot x_{w,G}, \qquad z_c = R_O \cdot x_{c,O} + R_G \cdot x_{c,G}, \quad c \in \{2, \ldots, n'_c\}, \tag{3.19}$$

User specifies the functions $K_c = K_c(p,T)$, $c \in \{1, \ldots, n'_c\}$:

$$x_{w,G} = K_w(p,T) x_{w,W}, \qquad x_{c,G} = K_c(p,T) x_{c,O}, \quad c \in \{2, \ldots, n'_c\}$$

From the equations (3.19) we obtain for each $c \in \{2, \ldots, n'_c\}$:

$$x_{w,G} = K_w, \quad x_{c,O} = z_c \frac{1}{R_O + K_c R_G}, \quad x_{c,G} = z_c \frac{K_c}{R_O + K_c R_G},$$

From the equilibrium conditions:

$$\sum_{P=1}^{n'_P} R_P = 1, \qquad \sum_{c=1}^{n'_c} x_{c,P} = 1$$

we obtain

$$R_W = z_w - R_G \cdot K_w \quad R_O = (1 - z_w) - R_G(1 - K_w) \quad x_{w,G} = K_w, \tag{3.20}$$

$$x_{c,O} = z_c \frac{1}{(1 - z_w) + R_G(K_c + K_w - 1)}, \quad x_{c,G} = z_c \frac{K_c}{(1 - z_w) + R_G(K_c + K_w - 1)}, \tag{3.21}$$

and $R_G \in [0, 1]$ is the solution of the equation

$$F(R_G) = 0, \qquad F(R_G) = \sum_{c=2}^{n'_c} \frac{z_c \cdot (K_c + K_w - 1)}{(1 - z_w) + R_G(K_c + K_w - 1)}$$

The values $K_i = K_i(p, T)$ can be specified via the tables KVTEMP (see 6.8.5), KVTABTn (see 6.8.6), KVTABLIM (see 6.8.7) in e300 (KVTABLIM (see 8.4.15), KVTABLE (see 8.4.16) in stars), or via the correlation formula:

$$K_i(p, T) = (A_i + B_i/p + C_i p) \cdot e^{-D_i/(T - E_i)} \tag{3.22}$$

where the coefficients A_i, B_i, C_i, D_i, E_i are set using the keywords KVCR (see 6.8.4) in e300 (KV1 / KV2 / KV3 / KV4 / KV5 (see 8.4.17) in stars). In stars data format this formula is also used for water. In e300 data format the following correlation is used

$$K_w(p, T) = \frac{10}{p} \cdot \left(\frac{T_C}{a}\right)^{1/b}, \quad a = 180.89, \quad b = 0.2350, \quad T_C(°C) = T(°K) - 273.15 \tag{3.23}$$

It is very difficult to choose many parameters in (3.22) ($K_i(p, T)$ should be positive and increasing in conjunction with T). In e300 data format the correlation (Wilson) for hydrocarbon components can be used – KVWI (see 6.8.8):

$$K_i(p, T) = \frac{p_{ci}}{p} \cdot e^{5.372697 \cdot (1 + A_i) \cdot (1 - T_{ci}/T)} \tag{3.24}$$

where

- T_{ci} – component critical temperature TCRIT (see 6.7.11);

- p_{ci} – component critical pressure PCRIT (see 6.7.12);

- A_i – component acentric factor ACF (see 6.7.15).

3.10 Phase saturations

Since (3.18) the number of moles of the "mobile" phases in the volume V_f is $M_P \cdot V_f$, and their volume is equal to $M_P \cdot V_f / \xi_P$. From the equation (3.1) we obtain $S_P = V_P / V_f = M_P / \xi_P$ or

$$S_P = \frac{N_{tot} \cdot R_P}{\xi_P}, \qquad P = 1, \ldots, n_P'$$

(3.25)

The description of calculations of the solid phase saturation is in the section (3.4).

3.11 Water viscosity

Water viscosity can be specified as a function of temperature using the tables – WATVISCT (see 6.8.35) (e300 data format), VISCTABLE (see 8.4.38) (stars data format), or via correlations. In e300 the following correlation is used (Grabovski):

$$\mu_W(T) = 1 / \left(A_W + B_W T_C + C_W T_C^2 \right), \quad A_W = 0.1323, \ B_W = 0.03333, \ C_W = 7.643 \cdot 10^{-6}$$

(3.26)

where T_C – the temperature $^\circ C$. In stars the following correlation is used:

$$\mu_W(T) = A_W \exp\left(B_W / T \right)$$

(3.27)

where T – the temperature $^\circ K$, the coefficients A_W, B_W are set via the keywords AVISC (see 8.4.36), BVISC (see 8.4.37).

Multiple viscosity regions can be specified via VSTYPE (see 8.4.34), VISCTYPE (see 8.4.35) (stars). tNavigator also uses the keyword (VISCNUM (see 6.4.9)).

In e300 data format the viscosity dependence on pressure can be specified:

$$\mu_W(p, T) = \mu_W(T) \frac{\mu_W'(p)}{\mu_W'(p_{ref})}$$

(3.28)

where

- $\mu_W(T)$ – water viscosity (depends on temperature) is calculated above;

- $\mu_W'(p)$ – water viscosity as a function ot pressure (specified using PVTW (see 6.5.4));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

3.12 Oil viscosity

Oil viscosity is calculated using the formula

$$\mu_O(T) = \prod_{k=2}^{n_c'} (\mu_{k,O})^{f_k(x_{k,o})}$$

(3.29)

where **oil component viscosity** $\mu_{k,O}(T)$ can be specified as a function of temperature using the tables of this keyword OILVISCT (see 6.8.36) (in e300 data format), VISCTABLE (see 8.4.38) (stars), or using correlation (OILVISCC (see 6.8.37)).

$f_k(x)$ (default: $f_k(x) = x$) are specified using OILVINDX (see 6.8.38) (in e300 data format) or VSMIXCOMP (see 8.4.39), VSMIXENDP (see 8.4.40), VSMIXFUNC (see 8.4.41) (stars). tNavigator also uses the keyword OILVINDT (see 6.8.39).

In e300 data format the following correlations can be used to calculate $\mu_{k,O}$ (specified using this keyword OILVISCC):

Name	Formula	Name	Formula
ASTM	$\log_{10}(\mu_{k,O} + A_k) = B_k T^{C_k}$	Vogel	$\log_{10}(\mu_{k,O}) = A_k + B_k/(T + C_k)$
Andrade	$\log_{10}(\mu_{k,O}) = A_k + B_k/T$	logarithmic	$\log_{10}(\mu_{k,O}) = A_k + B_k \log_{10}(T)$

In stars data format Andrade correlation is used with parameters $A_k = \log_{10} A'_k$, $B_k = B'_k \log_{10} e$, where A'_k, B'_k are specified using keywords AVISC (see 8.4.36), BVISC (see 8.4.37).

Multiple viscosity regions can be specified via VSTYPE (see 8.4.34), VISCTYPE (see 8.4.35) (stars). tNavigator also uses the keyword (VISCNUM (see 6.4.9)).

In e300 data format pressure dependence can be set:

$$\mu_O(p, T) = \mu_O(T) \frac{\mu'_O(p)}{\mu'_O(p_{ref})} \tag{3.30}$$

where

- $\mu_O(T)$ – oil viscosity (depends on temperature);

- $\mu'_O(p)$ – oil viscosity as a function of pressure (specified using PVCO (see 6.5.5));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

3.13 Gas viscosity

In e300 data format **gas viscosity** is calculated

$$\mu_G(p, T) = \sum_{k=1}^{n'_c} x_{k,G} \mu_{k,G} \tag{3.31}$$

component water viscosity (water vapor) $\mu_{1,G}(p, T)$ in the gas phase

$$\mu_{1,G}(p, T) = A_g + B_g T_C + C_g (p/10)^{D_g}, \quad T_C = T - 273.15$$
$$A_g = 4.9402 \cdot 10^{-3}, \quad B_g = 5.0956 \cdot 10^{-5}, \quad C_g = 2.9223 \cdot 10^{-6}, \quad D_g = 2.5077$$

hydrocarbon component viscosity $\mu_{k,G}(T)$ in the gas phase can be specified as a function of temperature using the tables of this keyword GASVISCT (see 6.8.40), or using the correlation formula with the coefficients GASVISCF (see 6.8.41):

$$\mu_{k,G}(T) = A_k \cdot T^{B_k} \tag{3.32}$$

In stars data format **gas viscosity** is calculated as

$$\mu_G(T) = \left(\sum_{k=1}^{n'_c} \mu_{k,G} \cdot x_{k,G} \sqrt{MW_k} \right) \Big/ \left(\sum_{k=1}^{n'_c} x_{k,G} \sqrt{MW_k} \right) \tag{3.33}$$

where MW_k – molecular weight of the component k (specified using CMM (see 8.4.44)), component viscosities are set via correlation (3.32) with the coefficients AVG (see 8.4.42), BVG (see 8.4.43). If gas viscosity isn't specified by user then in stars data format it is calculated as

$$\mu_G(T) = 0.0136 + 3.8 \cdot 10^{-5} \cdot T_C, \quad T_C = T - 273.15$$

Multiple viscosity regions can be specified via VSTYPE (see 8.4.34), VISCTYPE (see 8.4.35) (stars). tNavigator also uses the keyword (VISCNUM (see 6.4.9)).

3.14   Enthalpy and heat capacity of the components

Thermodynamic properties of the component $c = 1, \ldots, n_c$ in the phases $P = W, O, G, S$:

- $H_{c,P}(T)$ – **enthalpy** of the component c in the phase P;

- $CP_{c,P} = dH_{c,P}(T)/dT$ – **heat capacity** of the component c in the phase P;

- HV_c – **vaporization enthalpy** of the component c (from liquid phase to the gas phase).

The following equations take place:

$$HV_c = H_{c,G} - (H_{c,W} + H_{c,O}) \tag{3.34}$$

so only 2 (of 3) sets of data should be specified $H_{c,O}$ (or $H_{c,W}$), $H_{c,G}$, HV_c. For each component only one parameter of $H_{c,O}$ or $H_{c,W}$ isn't zero, because there are no components which can be in the water and the oil phase at the same time.

Component enthalpy is specified via the heat capacity.
In e300 data format: $CP_{c,P}$ $(kJ/kg/°C)$. In stars data format: $CP_{c,P}$ $(J/mol/°C)$.

3.15   Enthalpy and internal energy of the phases

Enthalpy of the phase $H_P(T)$ (for one mole, kJ/mol):

$$H_P(p,T) = \sum_{c=1}^{n_c} x_{c,P}(p,T) \cdot H_{c,P} \cdot MW_c \quad \text{(in e300)} \quad \text{or} \quad H_P(p,T) = \sum_{c=1}^{n_c} x_{c,P}(p,T) \cdot H_{c,P} \quad \text{(in stars)}$$

$$\tag{3.35}$$

where MW_c – molecular weight given by the keyword MW (see 6.7.14) (kg/mol). In e300 data format the gas phase component enthalpies include both a temperature dependent term and a pressure dependent term (Joule-Thomson) (see (3.44), (3.45)).

The internal energy of the phases (for unit volume, kJ/m^3) for "mobile" phases:

$$U_P(p,T) = \xi_P(p,T)(H_P(T) - p_{Pp}/\xi_P) = \xi_P(p,T) \cdot H_P(T) - p_{Pp}$$

where p_{Pp} – **the partial pressure** of the phase P, $P = W, O, G$. Then the internal energy (for unit volume, kJ/m^3) of the pore volume:

$$U_f(p,T) = \sum_{P=1}^{n'_P} \xi_P(p,T) \cdot H_P(T) - p \qquad (3.36)$$

The internal energy (for unit volume, kJ/m^3) of the volume which is filled with the solid phase:

$$U_S(p,T) = \xi_S(p,T) \cdot H_S(T) \qquad (3.37)$$

3.16 Water enthalpy

In e300 data format **the enthalpy**, **the water vaporization enthalpy** are taken from the internal tables (see Perry's Chemical Engineers' Handbook, p. 2-306). These enthalpies are the functions of pressure and temperature. In stars data format **the enthalpy**, **the water vaporization enthalpy** are also taken from the internal tables.

3.17 Liquid enthalpy

In e300 data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2 \qquad (3.38)$$

where the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHA (see 6.8.53), SPECHB (see 6.8.54) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).

In stars data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i \qquad (3.39)$$

where the coefficients $CP_{i,c}$, $i = 1, \ldots, 4$ are specified using the keywords CPL1 / CPL2 / CPL3 / CPL4 (see 8.4.18).
tNavigator also uses the keywords $CP_{3,c}$ = SPECHC (see 6.8.55), $CP_{4,c}$ = SPECHD (see 6.8.56).
Default values: $CP_{1,c} = 0.5 Btu/lbmol/F = 0.5 * 1.05506/0.453592 * 1.8 kJ/mol/C = 2.0934 kJ/mol/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).
Then the liquid phase enthalpy is calculated according to (3.35).

3.18 Vaporization enthalpy

The enthalpy of a gaseous oil component is calculated as

$$HV_c(T) = A_c \cdot (1 - T/T_{c,crit})^{B_c} = A'_c (T_{c,crit} - T)^{B_c}, \quad A'_c = A_c/T_{c,crit}^{B_c} \tag{3.40}$$

where:
In e300 data format A_c is specified using the keyword HEATVAP (see 6.8.12) (default: $0 kJ/kg$).
In stars data format A'_c is specified using the keyword HVR (see 8.4.21) (default: $0.25 Btu/lbmol/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/mol/C = 1.0467 kJ/mol/C$).
B_c (default: 0.38) is specified using the keyword HEATVAPE (see 6.8.13) (e300), EV (see 8.4.22) (stars).
Critical temperature of the component $T_{c,crit}$ is specified using the keyword TCRIT (see 6.7.11) (e300), TCRIT (see 8.4.12) (stars).
If $T \geq T_{c,crit}$ then $HV_c(T) = 0$.

If the component enthalpy in the liquid phase and vaporization enthalpy is specified then the gas phase enthalpy is calculated from the equation (3.34).

If the component enthalpy in the gas phase and vaporization enthalpy is specified then the oil phase enthalpy is calculated from the equation (3.34).

3.19 Gas phase enthalpy

In e300 data format **water enthalpy in the gas phase** is calculated as

$$H_{1,G}(T) = h_c + RT_c \left((1 - T_r)(A + BT_r^{\beta}) + C(1 - T_r)^{\alpha} + D(1 - T_r)^{2\alpha} \right), \tag{3.41}$$

where $T_r = \dfrac{T}{T_c}$ и

$$
\begin{array}{lll}
\alpha = 0.2866, & \beta = 3.140, & T_c = 647.126K, \\
R = 0.461522 kJ/kgK, & h_c = 2086 kJ/kg, & A = -7.818955, \\
B = 3.977657, & C = 2.665298, & D = 4.754665,
\end{array}
$$

If $c \geqslant 2$ (enthalpy of hydrocarbon components)

$$H_{c,G}(T) = h_{c,G} + CP_{1,c}(T - T_{ref}) + \frac{1}{2} CP_{2,c}(T - T_{ref})^2, \tag{3.42}$$

where the coefficients $h_{c,G}$ (kJ/kg), $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ ($kJ/kg/°C/°C$), are specified using the keywords HEATVAPS (see 6.8.61), SPECHG (see 6.8.57), SPECHH (see 6.8.58) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).

In stars data format **the component gas phase enthalpy** is calculated as

$$H_{c,G}(T) = h_{c,G} + \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i \tag{3.43}$$

where the coefficients $h_{c,G}$, $CP_{i,c}$, $i = 1,\ldots,4$ are specified using the keywords HVAPR (see 8.4.20), CPG1 / CPG2 / CPG3 / CPG4 (see 8.4.19).

tNavigator also uses the keywords $CP_{3,c}$ = SPECHI (see 6.8.59), $CP_{4,c}$ = SPECHJ (see 6.8.60). Default values: $h_{c,G} = 0.25 Btu/lb/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/kg/C = 1.0467 kJ/kg/C$, $CP_{1,c} = 0.25 Btu/lb/F = 1.0467 kJ/kg/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).

Then the gas phase enthalpy is calculated according to (3.35).

If the vaporization enthalpy isn't specified, then during e300 data format instead of (3.35) is used:

$$H_G(p,T) = \sum_{c=1}^{n_c} x_{c,G}(p,T) \cdot \big(H_{c,G}(T) \cdot MW_c + H_{JT,c}(p) \big) \qquad (3.44)$$

where (Joule-Thomson)

$$H_{JT,c}(p) = -10^2 \cdot Z_{c,1} \cdot (p - p_{ref}) \qquad (3.45)$$

The coefficient $Z_{k,1}$ is set via ZFACT1 (see 6.8.33), p_{ref} – STCOND (see 6.7.7). Multiplier 10^2 (3.45) results from units transformation: $Z_{k,1}$ – $m^3/kg-mol$, so $Z_{c,1} \cdot (p - p_{ref})$ – $m^3/kg-mol \cdot bar = 10^5 \cdot m^3 \cdot Pa/kg-mol = 10^2 \cdot kJ/kg-mol$.

3.20 Solid phase enthalpy

The component solid phase enthalpy is calculated as

$$H_{c,S}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2} CP_{2,c}(T - T_{ref})^2 \qquad (3.46)$$

where

- in e300 data format the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHS (see 6.8.62), SPECHT (see 6.8.63) (default: 0), T_{ref} is specified using the keyword STCOND (see 6.7.7).

- in stars data format the coefficients $CP_{1,c}$, $CP_{2,c}$ are specified using the keywords SOLID_CP (see 8.4.14) (default: $CP_{1,c} = 17 kJ/mol/C$, $CP_{2,c} = 0$), T_{ref} is specified using the keyword TEMR (see 8.4.4).

Then the solid phase enthalpy is calculated according to (3.35).

3.21 Rock enthalpy

Rock enthalpy (for unit volume, kJ/m^3) is calculated as

$$H_R(T) = \big(CP_1(T - T_{ref}) + \frac{1}{2} CP_2(T - T_{ref})^2 \big) \qquad (3.47)$$

where

- in e300 data format the coefficients CP_1 ($kJ/m^3/°C$), CP_2 are set via HEATCR (see 6.8.9), HEATCRT (see 6.8.10) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).

- in stars data format the coefficients CP_1, CP_2 are set via ROCKCP (see 8.3.3), (default $CP_1 = 2347kJ/m^3$, $CP_2 = 0$), T_{ref} is specified using TEMR (see 8.4.4).

In tNavigator the coefficients CP_1, CP_2 can be specified via the keyword HEATTCR (see 6.8.11).

Internal rock energy (for unit volume, kJ/m^3):

$$U_R(T) = H_R(T) \tag{3.48}$$

3.22 Block internal energy

Block internal energy (kJ) is calculated: the internal energies of the "mobile" phases (3.36), solid phase (3.37) and rock (3.48), (in the unit volume) are multiplied by the volume (see 45 and (3.2)):

$$
\begin{aligned}
U_b(p,T) &= U_f(p,T) \cdot V_f + U_S(p,T) \cdot V_S + U_R(T) \cdot V_R \\
&= V_b \cdot \phi \left(1 - \widehat{S}_S\right) \left(\sum_{P=1}^{n'_P} \xi_P(p,T) \cdot H_P(T) - p \right) \\
&\quad + V_b \cdot \phi \cdot \widehat{S}_S \cdot \xi_S(p,T) \cdot H_S(T) + V_b \cdot (1 - \phi) \cdot H_R(T)
\end{aligned} \tag{3.49}
$$

3.23 Porosity

In e300 data format **the porosity** $\phi = \phi(p,x,y,z)$ is

$$\phi(p,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c(p - p_{ref}) + c^2(p - p_{ref})^2/2) \tag{3.50}$$

where

- $\psi(x,y,z)$ – **net to gross values** (NTG (see 6.2.18));

- $\phi(x,y,z)$ – porosity at the pressure p_{ref} (PORO (see 6.2.17));

- c – compressibility (ROCK (see 6.5.12)) (default: in e100 $= 0$, in e300 $= 4.934 \cdot 10^{-5}/Bar$);

- p_{ref} – reference pressure for $\phi(x,y,z)$ (ROCK (see 6.5.12)) (default $1.0132Bar$).

In stars data format **the porosity** $\phi = \phi(p,T,x,y,z)$ is

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref})) \tag{3.51}$$

where

- $\psi(x,y,z)$ – **net to gross value** (NETPAY (see 8.2.2));

- $\phi(x,y,z)$ – porosity at the pressure p_{ref} (POR (see 7.3.7));

- c_p, c_T, c_{pT} – compressibilities (CPOR (see 8.3.5), CTPOR (see 8.3.6), CPTPOR (see 8.3.7)) (default: 0); tNavigator also uses the keyword ROCKT (see 6.8.16) to specify compressibilities c_T, c_{pT};

- p_{ref} – reference pressure for $\phi(x,y,z)$ (PRPOR (see 8.3.4)) (default – pressure at the first active grid block).

- T_{ref} – reference temperature given by the keyword TEMR (see 8.4.4).

3.24   Pore volume of grid block

In e300 data format pore volume $V_{p,ref}$ at reference pressure p_{ref} (see 3.23) is:

- If PORV (see 6.2.20) is specified, then $V_{p,ref} =$ PORV (see 6.2.20)

- Else
$$V_{p,ref}(x,y,z) = \gamma(x,y,z)\,\psi(x,y,z)\,\phi(x,y,z)V_{geom} \qquad (3.52)$$
where a multiplier $\gamma(x,y,z)$ (default: 1) is specified via MULTPV (see 6.2.21), V_{geom} — geometric block volume.

Then **pore volume**

$$V_p(x,y,z) = V_{p,ref}(x,y,z)(1 + c(p - p_{ref}) + c^2(p - p_{ref})^2/2) \qquad (3.53)$$

see 3.23.

In stars data format pore volume $V_{p,ref}$ at reference pressure p_{ref} and temperature T_{ref} (see 3.23) is calculated via formula (3.52), where multiplier $\gamma(x,y,z)$ (default: 1) is set via VOLMOD (see 8.2.1). Then **pore volume**

$$V_p(x,y,z) = V_{p,ref}(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref})) \quad (3.54)$$

see 3.23.

3.25   Bulk volume of grid block

In e300 data format bulk volume of grid block V_b:

- Tf ROCKV (see 6.8.68) is specified, then $V_b = V_{p,ref} +$ ROCKV (see 6.8.68).

- Else $V_b = \max\{V_{geom}, V_{p,ref}\}$.

In stars data format bulk volume of grid block $V_b = \max\{\gamma \cdot V_{geom}, V_{p,ref}\}$ where a multiplier γ (default: 1) is specified via VOLMOD (see 8.2.1).

3.26 Bulk volume of rock

Bulk volume of rock V_R:

$$V_R = \max\{V_b - V_p, 0\}.$$

3.27 Thermal conductivity of the grid block

In e300 data format **thermal conductivity of the grid block** is

$$K_b = (1 - \alpha S_G) \cdot k_R \tag{3.55}$$

where

- k_R – rock thermal conductivity (THCONR (see 6.8.14)) ($kJ/m/day/^\circ C$);

- α – is set via THCONSF (see 6.8.15), $\alpha \in [0, 1]$ (default: 0);

- S_G – gas saturation.

In stars data format **thermal conductivity of the grid block** is specified using the keyword THCONMIX (see 8.3.13) option SIMPLE

$$K_b = \phi \left(1 - \widehat{S}_S\right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R \tag{3.56}$$

where

- k_P, $P = W, O, G, S$ – phase thermal conductivity (THCONW (see 8.3.9), THCONO (see 8.3.10), THCONG (see 8.3.11), THCONS (see 8.3.12)) (default THCONS (see 8.3.12) = THCONR (see 8.3.8));

- S_P, $P = W, O, G$ – phase saturation, $\widehat{S}_S$ – solid phase saturation;

- k_R – rock thermal conductivity (THCONR (see 8.3.8)) (default $149.6 kJ/m/day/C$);

- ϕ – porosity.

tNavigator also uses the keyword THCONT (see 6.8.17) to specify the parameters k_P, $P = W, O, G, S$ и k_R.

In stars data format isothermal part of **the block thermal conductivity** specified using the keyword THCONMIX (see 8.3.13) option COMPLEX

$$K_b' = \left(1 - \sqrt{S_W + S_O}\right) \cdot k_G \cdot F\left(\frac{k_R}{k_G}\right) + \sqrt{S_W + S_O} \cdot k_L \cdot F\left(\frac{k_R}{k_L}\right)$$

where

$$F(x) = \exp\left((0.28 - 0.32876 \cdot \log \phi_f - 0.024755 \cdot \log x) \log x\right), \quad K_L = \frac{k_W S_W + K_O S_O}{S_W + S_O}$$

where

- ϕ_f – "mobile" porosity.

The dependence between **the block thermal conductivity** and the temperature

$$K_b = K_b' - 1.7524 \cdot 10^{-5}(T - T_{ref}) \cdot (K_b' - 119616)$$
$$\cdot \left(K_b'\right)^{-0.64} \cdot \left(K_b' \cdot \left(1.8 \cdot 10^{-3} \cdot T\right)^{\left(-3.6784 \cdot 10^{-6} K_b'\right)} + 110644.8\right)$$

where T_{ref} is given by the keyword TEMR (see 8.4.4).

3.28 Chemical reactions

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE (see 6.8.42) in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT (see 6.8.43) in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD (see 6.8.44), REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

For example, for $n_c = 5$, components $C_{12}H_{26}$, C_3H_8, O_2, CO_2, H_2O and $n_r = 2$ reactions:

$$C_{12}H_{26} + 18.5O_2 \rightarrow 12CO_2 + 13H_2O$$
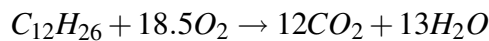
	$C_{12}H_{26}$	C_3H_8	O_2	CO_2	H_2O
S_{Rr}	1	0	18.5	0	0
S_{Pr}	0	0	0	12	13
N_r	1	0	1	0	0

$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$
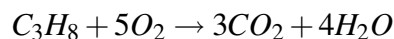
	$C_{12}H_{26}$	C_3H_8	O_2	CO_2	H_2O
S_{Rr}	0	1	5	0	0
S_{Pr}	0	0	0	3	4
N_r	0	1	1	0	0

Reaction rate of the reaction number r, $r = 1, \ldots, n_r$ ($kg - mol/day$) for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c} (c'_{ri})^{n_{ri}} \tag{3.57}$$

where $R = 8.3143 \dfrac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$c_{ri} = \begin{cases} \left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\ \widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\ N_i & \text{if the reactant } i \text{ is in all phases} \end{cases} \tag{3.58}$$

$$c'_{ri} = \begin{cases} \phi \cdot c_{ri} & \text{if it isn't gas and not REACPHA (see 6.8.50) with GPP (there is no O2PP (see 8.4.33))} \\ p' \cdot x_{i,G} & \text{else (for gas)} \end{cases} \tag{3.59}$$

where temperature, pressure and order are:

$$T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases} \quad p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases} \quad n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases} \tag{3.60}$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

The component phase in the chemical reaction i (see the description of c_{ri} above), is set via REACPHA (see 6.8.50) in e300, RPHASE (see 8.4.29) in stars. In e300 data format the "mobile" component can be specified in **all phases**, then $c_{ri} = \phi \cdot N_i$. This is a default value. In stars data format the default value is calculated using the data of the keyword MODEL (see 8.4.1).

The reaction rate (3.57) can depend on pore volume $V_p = \phi \cdot V_b$:

$$R_r = V_p \cdot \phi^{n_{r,p}} \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i \in F_r}(c_{ri})^{n_{ri}} \cdot \prod_{i \notin F_r}(p' \cdot x_{i,G})^{n_{ri}} \tag{3.61}$$

where $n_{r,p} = \sum\limits_{i \in F_r} n_{ri} - 1$, indices F_r:

$F_r = i \in \{1, \ldots, n_c : $ if i is no gas and there is no REACPHA (see 6.8.50) with GPP (no O2PP (see 8.4.33)

In e300 data format another value $n_{r,p}$ can be specified for each reaction using REACPORD (see 6.8.47). The reaction rate can be independent of pore volume ($n_{r,p} = 0$).

The additional **component flow** i, $i = 1, \ldots, n_c$, appears via the chemical reactions:

$$Q_{Ri} = \sum_{r=1}^{n_r} (S_{Pri} - S_{Rri})R_r. \tag{3.62}$$

The additional **energy flow**, appears via the chemical reactions:

$$Q_{Re} = \sum_{r=1}^{n_r} H_r R_r. \tag{3.63}$$

3.29   The heat exchange between the reservoir and surroundings

In e300 and stars data format there is the same modelling of the **heat exchange between the reservoir and surroundings**: "A Simple Method for Predicting Cap and Base Rock Heat Losses in Thermal Reservoir Simulators Vinsome, P.K.W., Westerveld, J.D., The Journal of Canadian Petroleum Technology (JCPT), (Montreal), July-September 1980, Volume 19, No. 3, 87-90.

The following parameters are to be specified:

- The connection between the reservoir and cap and base rocks. In e300 data format the complicated form of connection can be specified via the keyword ROCKCON (see 6.2.58), in stars – the connection can be: the entire surface of the rock region (THTYPE (see 8.3.2), ROCKTYPE (see 8.3.1)) in the given direction, – specified via the keyword HLOSSPROP (see 8.3.16). tNavigator also uses the keyword ROCKCONT (see 6.2.59), which specifies the heat loss directions, volumetric heat capacity and rock conductivity.

- The number of types of cap and base rocks with different properties, which will be used to model the heat exchange between the reservoir and surroundings in e300 is set via ROCKDIMS (see 6.1.20), in stars – is specified at the same time with the description of the geometry.

- Volumetric heat capacity ($kJ/m^3/C$) – ROCKPROP (see 6.2.57) (e300), HLOSSPROP (see 8.3.16) (stars).

- Rock conductivity ($kJ/m/day/C$) – ROCKPROP (see 6.2.57) (e300), HLOSSPROP (see 8.3.16) (stars).

- Initial temperature (C) – ROCKPROP (see 6.2.57) (e300), HLOSST (see 8.3.14) (stars).

- Temperature-dependent coefficient of the volumetric heat capacity of the rock ($kJ/m^3/C^2$) – ROCKPROP (see 6.2.57) in e300 (is disable in stars).

- Minimal difference between temperatures when the calculations of the heat exchange should start (C) – HLOSSTDIFF (see 8.3.15) in stars (dafault 0) (is disable in e300).

Using these parameters tNavigator calculates for each grid block the value Q_L – **the energy of heat exchange between the reservoir and surroundings** ($Q_L = 0$ – if the block isn't situated on the the reservoir boundary).

3.30 Phase flow rate

$$u_p = -\beta_c \left(\boldsymbol{k} \frac{k_{rP}}{\mu_P} (\nabla p + \nabla P_{cP} - \rho_P g \nabla d) \right) \qquad (3.64)$$

where

- β_c – the constant value, g – gravitation constant,

- $k_{rP} = k_{rP}(S_W, S_G)$ – phase relative permeability,

- $d = d(x, y, z)$ – depth (top-down).

3.31 Mass conservation equation

Mass conservation equation (moles) for each component:

$$\frac{\partial}{\partial t} \left(\phi(1 - \widehat{S}_S) N_c \right) = \text{div} \sum_{P=1}^{n'_P} x_{c,P} \xi_P U_P + Q_c + Q_{Rc}, \quad c = 1, \dots, n'_c \qquad (3.65)$$

$$\frac{\partial}{\partial t} (\phi N_c) = Q_{Rc}, \quad c = n'_c + 1, \dots, n_c \qquad (3.66)$$

where

- $U_P = U_P(p, \boldsymbol{N})$ – velocity vector (3.64) of phase flow P, $P = 1, \dots, n'_P$;

- $Q_c = \sum_{\beta=1}^{n_w} \delta_\beta q_c^\beta$ — total rate of all sources and flows; δ_β – δ-Dirack function, on the trajectory of source (flow) number β, q_c^β – rate (negative for flows and positive for sources), n_w – total number of sources and flows;

- Q_{Rc} – component c flow, appears via the chemical reactions (3.62);

- $\widehat{S}_S$ – solid phase saturation (3.4);

- n'_P, n'_c – see page 47.

Sources and flows — **injectors and producers** and **aquifers**.

3.32 Energy conservation equation

Energy conservation equation:

$$\frac{\partial}{\partial t}(U_b) = F_e + C_e + Q_{Re} + Q_e^{well} - Q_L \tag{3.67}$$

where

- **internal block energy** (kJ) U_b specified via (3.49);

- **convection enthalpy flow** from (in) neighbouring surrounding points

$$F_e = \operatorname{div} \sum_{P=1}^{n'_P} H_P \xi_P U_P$$

  where **phase enthalpy** H_P specified via (3.35), $U_P = U_P(p, \boldsymbol{N})$ – velocity vector (3.64) of phase flow;

- energy flow due to **conductivity**

$$C_e = \operatorname{div}(K_b \nabla T),$$

  where K_b — **block thermal conductivity** $(kJ/m/day/°C)$, see page 62;

- energy flow due to **chemical reactions** Q_{Re} see (3.63);

- **energy flow from the well**

$$Q_e^{well} = \sum_{P=1}^{n'_P} H_P \cdot \xi_P \cdot Q_P, \tag{3.68}$$

  where Q_P – rate of phase P for connection in the grid block, **phase enthalpy** H_P specified via (3.35);

- Q_L — **energy flow** due to heat exchange with surrounding, see page 65.

Total **energy conservation equation** for block V_b:

$$\frac{\partial}{\partial t}\left(V_P\left(1 - \widehat{S}_S\right)\left(\left(\sum_{c=1}^{n'_c} N_c\right)\sum_{P=1}^{n'_P}(R_P \cdot H_P) - 10^2 \cdot p\right) + V_P\left(\sum_{c=n'_c+1}^{n_c} N_c \cdot H_{c,S}(T)\right) + V_R \cdot H_R\right)$$

$$= \int\limits_{\partial V_b}\left(\sum_{P=1}^{n'_P} H_P \xi_P U_P\right) ds + \int\limits_{\partial V_b} K_b \nabla T \, ds + Q_{Re} + Q_e^{well} - Q_L.$$

Where $V_R = \max\{V_b - V_p, 0\}$.

3.33 Initial conditions

For the equations (3.65), (3.67) one should specify initial p, N_c, T. tNavigator uses two variants of specification of initial conditions:

- explicit specification of initial conditions;

- initial conditions are calculated from hydrostatic and thermodynamic equilibrium conditions.

3.33.1 Explicit specification of initial conditions

In this case the following parameters are always specified explicitly:

- **pressure** p – via PRESSURE (see 6.9.7) (PRES (see 8.6.10));

- **temperature** T – via TEMPI (see 6.9.17) (TEMP (see 8.6.11)).

For solid phase always

- in e300 data format one should specify explicitly **solid phase saturation** $\widehat{S}_S$ and component distribution $x_{c,S}$ (see (3.4) via SSOLID (see 6.9.12) and SMF (see 6.9.13); then molar density of solid phase is calculated (3.17) and molar densities N_c, $c = n'_c + 1, \ldots, n_c$;

- in stars data format one should specify explicitly **molar densities** N_c, $c = n'_c + 1, \ldots, n_c$ of components, which are present in solid phase, via CONC_SLD (see 8.6.12).

Phase and component composition of **'mobile' phases** and components can be specified several ways.

- tNavigator (this option is disable in stars and e300 data formats) allows to specify explicitly **values** $z_c = N_c/N_{tot}$, $c = 1, \ldots, n'_c$, $N_{tot} = \sum\limits_{c=1}^{n'_c} N_c$ via ZMF (see 6.9.16). So **initialization algirithm** is the following:

  – calculation of $K_i = K_i(p, T)$ using pressure and temperature from (3.22), (3.23), (3.24)

  – calculation of the solution of equation (3.19) and values R_P, $x_{c,P}$

  – calculation of molar phase densities of 'mobile' phases ξ_P, $P = 1, \ldots, n'_P$ from (3.5), (3.7), (3.8), (3.11), (3.12), (3.13)

  – calculation of N_{tot}

  – calculation of molar densities of 'mobile' components $N_c = z_c \cdot N_{tot}$, $c = 1, \ldots, n'_c$.

- In e300 data format (this option is disable in stars) **value of** $z'_c = N_c/N'_{tot}$, $c = 2, \ldots, n'_c$, $N'_{tot} = \sum\limits_{c=2}^{n'_c} N_c$ is specified explicitly via ZMF (see 6.9.16), water saturation S_W is set via SWAT (see 6.9.9). Free gas is disable: $S_G = 0$, $S_O = 1 - S_W$, two keywords SOIL (see 6.9.11) and SGAS (see 6.9.10) are ignored. **Initialization algorithm** is the following:

– calculation of the matrix $x_{c,P}$:

$$x_{w,W} = 1, \; x_{c,W} = 0, \quad x_{w,O} = 0, \; x_{c,O} = z'_c \quad x_{w,G} = 0, \; x_{c,G} = 0, \quad c = 1, \ldots, n'_c;$$

– calculation of molar densities ξ_W, ξ_O;

– calculation of the values N_c:

$$N_w = \xi_W \cdot S_W, \quad N_c = x_{c,O} \cdot \xi_O \cdot S_O = z'_c \cdot \xi_O \cdot S_O, \quad c = 2, \ldots, n'_c.$$

- **Saturation of 'mobile' phases** S_P is specified explicitly using SWAT (see 6.9.9) (SW (see 8.6.9)), SOIL (see 6.9.11) (SO (see 8.6.7)), SGAS (see 6.9.10) (SG (see 8.6.8)). **Consentration matix** $x_{c,P}$ $c = 1, \ldots, n'_c$, $P = O, G$ (for $P = W$) – using XMF (see 6.9.14) (MFRAC_OIL (see 8.6.13)), YMF (see 6.9.15) (MFRAC_GAS (see 8.6.14)). Since (3.1) one can specify only several saturations (not all), see a table below. S_{wc} – critical water saturation, which is calculated from the table of relative permeabilities at the given temperature.

N	S_W	S_O	S_G	E300 format	stars format	Calculations
1	√	√	√	√	√	
2	√	√		√	√	$S_G = 1 - S_W - S_O$
3	√		√	√	√	$S_O = 1 - S_W - S_G$
4		√	√	√	√	$S_W = 1 - S_O - S_G$
5	√			√	√	$S_O = 1 - S_W, S_G = 0$
6		√		√	√	$S_W = 1 - S_O, S_G = 0$
7			√		√	$S_W = S_{wc}, S_O = 1 - S_W - S_G$
8					√	$S_W = S_{wc}, S_G = 0, S_O = 1 - S_W$

Then **the initialization algorithm** is the following:

– calculation of molar densities of 'mobile' phases ξ_P, $P = 1, \ldots, n'_P$ from (3.5), (3.7), (3.8), (3.11), (3.12), (3.13)

– calculation of molar densities of 'mobile' components N_c, $c = 1, \ldots, n'_c$.

In `e300` and `stars` data formats initial conditions are specified in each grid block.

- Pressure – PRESSURE (see 6.9.7), PRES (see 8.6.10).

- Temperature – TEMPI (see 6.9.17), TEMP (see 8.6.11).

- Solid phase – SSOLID (see 6.9.12), SMF (see 6.9.13), CONC_SLD (see 8.6.12).

- 'Mobile' phases – ZMF (see 6.9.16), SWAT (see 6.9.9), SOIL (see 6.9.11), SGAS (see 6.9.10), XMF (see 6.9.14), YMF (see 6.9.15), SW (see 8.6.9), SO (see 8.6.7), SG (see 8.6.8), MFRAC_OIL (see 8.6.13), MFRAC_GAS (see 8.6.14).

3.33.2 Calculations of initial conditions from hydrostatic and thermodynamic equilibrium conditions

Solid phase saturation is specified **explicitly**, see 3.33.1.
Temperature T distribution:

- in e300 data format depends on depth – TEMPVD (see 6.8.64);

- in stars data format – is specified impicitly via TEMP (see 8.6.11).

In black oil case the pressure p and saturation of 'mobile' phases are calculated from hydrostatic equilibrium conditions – EQUIL (see 6.9.1) (VERTICAL (see 7.6.2)).

In these calculations **phase mass density** should be calculated. In black oil case one shouls specify PVT tables and the distribution of boiling points (dew points) versus depth.

In compositional run component composition of mixture should be specified in order to calculate phase mass densities. Stars supports only explicit specification of **concentration matrix** $x_{c,P}$ $c = 1, \ldots, n'_c$, $P = O, G$ in each block via MFRAC_OIL (see 8.6.13), MFRAC_GAS (see 8.6.14), see 3.33.1.

In e300 data format there are several ways to specify component composition of mixture versus depth.

- **Distrubution of values** $z_c = N_c/N_{tot}$, $c = 1, \ldots, n'_c$, $N_{tot} = \sum\limits_{c=1}^{n'_c} N_c$ versus depth is specified via ZMFVD (see 6.7.10). Then **mass density is calculated via the following algorithm**:

  - values $K_i = K_i(p, T)$ are calculated from pressure and temperature at selected depth (3.22), (3.23), (3.24);

  - solution of equations (3.19) and calculate the values R_P, $x_{c,P}$;

  - calculation of 'mobile' phases mass densities ρ_P, $P = 1, \ldots, n'_P$ from (3.5), (3.7), (3.8), (3.11), (3.12), (3.13).

  At the end of calculations:

  - calculated values z_c in grid blocks are saved in an array ZMF (see 6.9.16);

  - calculation of N_{tot};

  - calculation of 'mobile' components molar densities $N_c = z_c \cdot N_{tot}$, $c = 1, \ldots, n'_c$.

- Distribution of **concentration matrix** $x_{c,P}$ $c = 1, \ldots, n'_c$, $P = O, G$ (для $P = W$) versus pressure is set via XMFVP (see 6.7.8), YMFVP (see 6.7.9).

  Hence this initialization can lead to thermodynamic nonequilibrium distribution. Therefore only one $x_{c,P}$ (of the phase which saturation is > 0; priority is O, G) is used. The second $x_{c,P}$ is calculated.

  - if $S_O > 0$, then $x_{c,O} = \text{XMF}$, $x_{c,G} = K_c(p, T)x_{c,O}$, $c = 2, \ldots, n'_c$;

– if $S_G > 0$, then $x_{c,G} = \text{YMF}$, $x_{c,O} = x_{c,G}/K_c(p,T)$, $c = 2,\ldots,n_c'$.

'Mobile' phases mass densities ρ_P, $P = 1,\ldots,n_P'$ are calculated from (3.5), (3.7), (3.8), (3.11), (3.12), (3.13).

At the end of calculations:

– calculated values $x_{c,P}$ in grid blocks are saved in arrays XMF (see 6.9.14), YMF (see 6.9.15);

– calculation of 'mobile' phases molar densities ξ_P, $P = 1,\ldots,n_P'$ из (3.5), (3.7), (3.8), (3.11), (3.12), (3.13)

– calculation of 'mobile' components molar densities N_c, $c = 1,\ldots,n_c'$.

3.34 Phase relative permeabilites

Calculation of phase relative permeabilities contains the following stages:

1. Permeabilities and capillary pressure are calculated for two-phase systems water–oil and gas–oil (see the table 1).

2. Relative permeabilities (and capillary pressure) scaling for two-phase systems (see 3.34.2, see the table 3).

3. Oil relative permeability k_{rO} is calculated using the first or the second Stone's model.

3.34.1 Phase relative permeability for two-phase systems

Specification of phase relative permeabilities is in the table 1.

Value	e300 data format	CMG data format	Value description
$k_{rW}(S_W)$, $k_{rOW}(S_W)$	SWOF (see 6.6.1)	SWT (see 7.5.3)	relative permeability for two-phase system water–oil
$k_{rG}(S_G)$, $k_{rOG}(S_G)$	SGOF (see 6.6.2)	SLT (see 7.5.4)	relative permeability for two-phase system gas–oil
$P_{cOW}(S_W)$	SWOF (see 6.6.1)	SWT (see 7.5.3)	capillary pressure two-phase system water–oil
$P_{cOG}(S_G)$	SGOF (see 6.6.2)	SLT (see 7.5.4)	capillary pressure two-phase system gas–oil

Таблица 1: Phase relative permeabilities

Phase relative permeablities can be specified for different saturation regions (Saturation regions are speicified via SATNUM (see 6.4.2) (e300), KRTYPE (see 8.5.1) (CMG)). I - saturation region number for current block i. From the table 1 we take functions for I and enter for them constants from the table 2. We suppose that at initialization stage the following condition is checked

$$\max_{S_W} k_{rOW} = \max_{S_G} k_{rOG}$$

Value	Value description
S_{WL}	minimal value of S_W in table for water
S_{WCR}	maximal value of S_W in table for water, for which $k_{rW}(S_W) = 0$
S_{WU}	maximal value of S_W in table for water
S_{GL}	minimal value of S_G in table for gas
S_{GCR}	maximal value of S_G in table for gas, for which $k_{rG}(S_G) = 0$
S_{GU}	maximal value of S_G in table for gas
S_{OWCR}	maximal value of $S_O = 1 - S_W - S_{GL}$ in table for water, for which $k_{rOW}(S_W) = 0$
S_{OGCR}	maximal value of $S_O = 1 - S_G - S_{WL}$ in table for gas, for which $k_{rOG}(S_G) = 0$
k_{rWmax}	maximal value of function $k_{rW}(S_W)$
k_{rGmax}	maximal value of function $k_{rG}(S_G)$
k_{rOmax}	maximal value of function $k_{rOW}(S_W)$ и $k_{rOG}(S_G)$
k_{rWR}	$k_{rW}(1 - S_{OWCR} - S_{GL})$
k_{rGR}	$k_{rG}(1 - S_{OGCR} - S_{WL})$
k_{rORG}	$k_{rOG}(S_{GCR})$
k_{rORW}	$k_{rOW}(S_{WCR})$
P_{CGmax}	maximal value of function $P_{cOG}(S_G)$
P_{CWmax}	maximal value of function $P_{cOW}(S_W)$

Таблица 2: Phase relative permeablity constants

3.34.2 Phase relative permeablities scaling

Each constant in the table 2 can be changed,

- via phase permeabilities scaling;

- these constants can be spacified as temperature dependent constants;

see the table 3.

Phase relative permeability scaling is switched on using the keyword ENDSCALE (see 6.6.11) (e300 data format run).

In CMG data format initial conditions can be specified as constants in saturation regions (for example – SWR (see 8.5.3) etc.) and as constants in each grid block (for example – BSWR (see 8.5.4) etc.), in e300 data format – contant should be specified for each grid block. tNavigator supports all keyword from the table 3.

Calculated values should satisfy the following conditions:

1. $S'_{GU} \leqslant 1 - S'_{WL}$

2. $S'_{GL} \leqslant 1 - S'_{WU}$

3. $S'_{OWCR} + S'_{WCR} < 1$

4. $S'_{OGCR} + S'_{GCR} + S'_{WL} < 1$

Value	Number	e300 (replace "Number")	CMG (replace "Number")
S'_{WL}	S_{WL}	●SWL (see 6.6.14) ●ENPTVT (see 6.8.65), parameter 2	●BSWR (see 8.5.4) ●SWR (see 8.5.3) ●KRTEMTAB (see 8.5.2)
S'_{WCR}	S_{WCR}	●SWCR (see 6.6.16) ●ENPTVT (see 6.8.65), parameter 3	●BSWCRIT (see 8.5.6) ●SWCRIT (see 8.5.5) ●KRTEMTAB (see 8.5.2)
S'_{WU}	S_{WU}	●SWU (see 6.6.20) ●ENPTVT (see 6.8.65), parameter 4	●1 − BSOIRW (see 8.5.8) ●1 − SOIRW (see 8.5.7) ●1 − KRTEMTAB (see 8.5.2)
S'_{GL}	S_{GL}	●SGL (see 6.6.15) ●ENPTVT (see 6.8.65), parameter 5	●BSGCON (see 8.5.10) ●SGCON (see 8.5.9) ●KRTEMTAB (see 8.5.2)
S'_{GCR}	S_{GCR}	●SGCR (see 6.6.17) ●ENPTVT (see 6.8.65), parameter 6	●BSGR (see 8.5.12) ●SGR (see 8.5.11) ●KRTEMTAB (see 8.5.2)
S'_{GU}	S_{GU}	●SGU (see 6.6.21) ●ENPTVT (see 6.8.65), parameter 7	●1 − BSOIRG (see 8.5.14) ●1 − SOIRG (see 8.5.13) ●1 − KRTEMTAB (see 8.5.2)
S'_{OWCR}	S_{OWCR}	●SOWCR (see 6.6.18) ●ENPTVT (see 6.8.65), parameter 8	●BSORW (see 8.5.16) ●SORW (see 8.5.15) ●KRTEMTAB (see 8.5.2)
S'_{OGCR}	S_{OGCR}	●SOGCR (see 6.6.19) ●ENPTVT (see 6.8.65), parameter 9	●BSORG (see 8.5.18) ●SORG (see 8.5.17) ●KRTEMTAB (see 8.5.2)
k'_{rWmax}	k_{rWmax}	●KRW (see 6.6.26) ●ENKRVT (see 6.8.66), parameter 2	●BKRWIRO (see 8.5.20) ●KRWIRO (see 8.5.19) ●KRTEMTAB (see 8.5.2)
k'_{rGmax}	k_{rGmax}	●KRG (see 6.6.27) ●ENKRVT (see 6.8.66), parameter 3	●BKRGCW (see 8.5.22) ●KRGCW (see 8.5.21) ●KRTEMTAB (see 8.5.2)
k'_{rOmax}	k_{rOmax}	●KRO (see 6.6.25) ●ENKRVT (see 6.8.66), parameter 4	●BKROCW (see 8.5.24) ●KROCW (see 8.5.23) ●KRTEMTAB (see 8.5.2)
k'_{rWR}	k_{rWR}	●KRWR (see 6.6.26) ●ENKRVT (see 6.8.66), parameter 5	—
k'_{rGR}	k_{rGR}	●KRGR (see 6.6.27) ●ENKRVT (see 6.8.66), parameter 6	—
k'_{rORG}	k_{rORG}	●KRORG (see 6.6.25) ●ENKRVT (see 6.8.66), parameter 7	—
k'_{rORW}	k_{rORW}	●KRORW (see 6.6.25) ●ENKRVT (see 6.8.66), parameter 8	—
P'_{CGmax}	P_{CGmax}	●PCG (see 6.6.29) ●ENPCVT (see 6.8.67), parameter 2	●BPCGMAX (see 8.5.26) ●PCGEND (see 8.5.25) ●KRTEMTAB (see 8.5.2)
P'_{CWmax}	P_{CWmax}	●PCW (see 6.6.28) ●ENPCVT (see 6.8.67), parameter 3	●BPCWMAX (see 8.5.28) ●PCWEND (see 8.5.27) ●KRTEMTAB (see 8.5.2)

Таблица 3: Phase relative permeability scaling

Two-point phase relative permeability scaling keeps permeability values (from tables) in two points:

Permeability	Point 1	Point 2
k_{rW}	S'_{WCR}	S'_{WU}
k_{rG}	S'_{GCR}	S'_{GU}
k_{rOW}	S'_{WL}	$1 - S'_{OWCR} - S'_{GL}$
k_{rOG}	S'_{GL}	$1 - S'_{OGCR} - S'_{WL}$

For the given S_W, T:

$$S'_W(S_W, T) = \begin{cases} S_W & S'_{WCR}(T) \geqslant S'_{WU}(T) \\ S_{WCR} & S_W < S'_{WCR}(T) \\ S_{WU} & S_W > S'_{WU}(T) \\ S_{WCR} + (S_W - S'_{WCR}(T)) \dfrac{S_{WU} - S_{WCR}}{S'_{WU}(T) - S'_{WCR}(T)} & \text{else} \end{cases} \quad (3.69)$$

$$k'_{rW}(S_W, T) = \frac{k'_{rW max}(T)}{k_{rW max}} k_{rW}(S'_W(S_W, T)) \quad (3.70)$$

If there is an oil phase in the model, the following parameters are calculated

$$S'_{OW}(S_W, T) = \begin{cases} S_W & S'_{WL}(T) \geqslant 1 - S'_{OWCR}(T) - S'_{GL}(T) \\ S_{WL} & S_W < S'_{WL}(T) \\ 1 - S_{OWCR} - S_{GL} & S_W > 1 - S'_{OWCR}(T) - S'_{GL}(T) \\ S_{WL} + (S_W - S'_{WL}(T)) \times \\ \quad \times \dfrac{1 - S_{OWCR} - S_{GL} - S_{WL}}{1 - S'_{OWCR}(T) - S'_{GL}(T) - S'_{WL}(T)} & \text{else} \end{cases}$$

$$(3.71)$$

$$k'_{rOW}(S_W, T) = \frac{k'_{rO max}(T)}{k_{rO max}} k_{rOW}(S'_{OW}(S_W, T)) \quad (3.72)$$

$$S'_{cOW}(S_W, T) = \begin{cases} S_W & S'_{WL}(T) \geqslant S'_{WU}(T) \\ S_{WL} & S_W < S'_{WL}(T) \\ S_{WU} & S_W > S'_{WU}(T) \\ S_{WL} + (S_W - S'_{WL}(T)) \dfrac{S_{WU} - S_{WL}}{S'_{WU}(T) - S'_{WL}(T)} & \text{else} \end{cases} \quad (3.73)$$

$$P'_{cOW}(S_W, T) = \frac{P'_{CW max}(T)}{P_{CW max}} P_{cOW}(S'_{cOW}(S_W, T)) \quad (3.74)$$

If there is gas phase in the model, for the diven S_G, T:

$$S'_G(S_G,T) = \begin{cases} S_G & S'_{GCR}(T) \geqslant S'_{GU}(T) \\ S_{GCR} & S_G < S'_{GCR}(T) \\ S_{GU} & S_G > S'_{GU}(T) \\ S_{GCR} + (S_G - S'_{GCR}(T)) \dfrac{S_{GU} - S_{GCR}}{S'_{GU}(T) - S'_{GCR}(T)} & \text{else} \end{cases}$$ (3.75)

$$k'_{rG}(S_G,T) = \frac{k'_{rGmax}(T)}{k_{rGmax}} k_{rG}(S'_G(S_G,T))$$ (3.76)

If there is an oil phase in the model, the following parameters are calculated

$$S'_{OG}(S_G,T) = \begin{cases} S_G & S'_{GL}(T) \geqslant 1 - S'_{OGCR}(T) - S'_{WL}(T) \\ S_{GL} & S_G < S'_{GL}(T) \\ 1 - S_{OGCR} - S_{WL} & S_G > 1 - S'_{OGCR}(T) - S'_{WL}(T) \\ S_{GL} + (S_G - S'_{GL}(T)) \times \\ \quad \times \dfrac{1 - S_{OGCR} - S_{WL} - S_{GL}}{1 - S'_{OGCR}(T) - S'_{WL}(T) - S'_{GL}(T)} & \text{else} \end{cases}$$ (3.77)

$$k'_{rOG}(S_G,T) = \frac{k'_{rOmax}(T)}{k_{rOmax}} k_{rOG}(S'_{OG}(S_G,T))$$ (3.78)

$$S'_{cOG}(S_G,T) = \begin{cases} S_G & S'_{GL}(T) \geqslant S'_{GU}(T) \\ S_{GL} & S_G < S'_{GL}(T) \\ S_{GU} & S_G > S'_{GU}(T) \\ S_{GL} + (S_G - S'_{GL}(T)) \dfrac{S_{GU} - S_{GL}}{S'_{GU}(T) - S'_{GL}(T)} & \text{else} \end{cases}$$ (3.79)

$$P'_{cOG}(S_G,T) = \frac{P'_{CGmax}(T)}{P_{CGmax}} P_{cOG}(S'_{cOG}(S_G,T))$$ (3.80)

Phase relative permeabilities free-point scaling is enable in **e300** data format (see the keyword SCALECRS (see 6.6.13)). This method keeps permeability values in free points in the table below (additional point – S_{Wr} (S_{Gr})).

Permeability	Point 1	Point 2	Point 3
k_{rW}	S'_{WCR}	$1 - S'_{OWCR} - S'_{GL}$	S'_{WU}
k_{rG}	S'_{GCR}	$1 - S'_{OGCR} - S'_{WL}$	S'_{GU}
k_{rOW}	S'_{WL}	S'_{WCR}	$1 - S'_{OWCR} - S'_{GL}$
k_{rOG}	S'_{GL}	S'_{GCR}	$1 - S'_{OGCR} - S'_{WL}$

For the given S_W, T the following parameters are calculated:

$$S'_{Wr}(T) = \begin{cases} 1 - S'_{OWCR}(T) - S'_{GL}(T) & \text{if oil phase is enable} \\ 1 - S'_{GCR}(T) & \text{else} \end{cases}$$

$$S_{Wr} = \begin{cases} 1 - S_{OWCR} - S_{GL} & \text{if oil phase is enable} \\ 1 - S_{GCR} & \text{else} \end{cases}$$

If calculated value $S'_{Wr}(T)$ is not in an interval $[S'_{WCR}(T), S'_{WU}(T)]$ or $S'_{WCR}(T) \geqslant S'_{WU}(T)$, then $S'_W(S_W, T)$ is calculated via formula (3.69), else:

$$S'_W(S_W, T) = \begin{cases} S_{WCR} & S_W < S'_{WCR}(T) \\ S_{WCR} + (S_W - S'_{WCR}(T)) \dfrac{S_{Wr} - S_{WCR}}{S'_{Wr}(T) - S'_{WCR}(T)} & S'_{WCR}(T) \leqslant S_W < S'_{Wr}(T) \\ S_{Wr} + (S_W - S'_{Wr}(T)) \dfrac{S_{WU} - S_{Wr}}{S'_{WU}(T) - S'_{Wr}(T)} & S'_{Wr}(T) \leqslant S_W < S'_{WU}(T) \\ S_{WU} & S_W > S'_{WU}(T) \end{cases}$$
(3.81)

Note that $S'_W(S'_{Wr}(T), T) = S_{Wr}$.

If the keyword KRWR (see 6.6.26) is not specified or $k_{rW}(S_{Wr}) \geqslant k_{rWmax}$, then $k'_{rW}(S_W, T)$ is calculated via formula (3.70), else:

$$k'_{rW}(S_W, T) = \begin{cases} \dfrac{k'_{rWR}(T)}{k_{rW}(S_{Wr})} k_{rW}(S'_W(S_W, T)) & S_W < S'_{Wr}(T) \\ k'_{rWR}(T) + \dfrac{k'_{rWmax}(T) - k'_{rWR}(T)}{k_{rWmax} - k_{rW}(S_{Wr})} \times \\ \quad \times (k_{rW}(S'_W(S_W, T)) - k_{rW}(S_{Wr})) & S_W \geqslant S'_{Wr}(T) \end{cases}$$
(3.82)

Note that $k'_{rW}(S'_{Wr}(T), T) = k'_{rWR}(T)$.

If oil phase is present then $S'_{OW}(S_W, T)$, $k'_{rOW}(S_W, T)$ also are calculated. For the given S_W, T the following parameters are calculated:

$$S'_{OWr}(T) = S'_{WCR}(T), \quad S_{OWr} = S_{WCR}.$$

If calculated value $S'_{OWr}(T)$ is not in the interval $[S'_{WL}(T), 1 - S'_{OWCR}(T) - S'_{GL}(T)]$ or $S'_{WL}(T) \geqslant 1 - S'_{OWCR}(T) - S'_{GL}(T)$, then $S'_{OW}(S_W, T)$ is calculated via formula (3.71), else

$$
S'_{OW}(S_W, T) =
\begin{cases}
S_{WL} & S_W < S'_{WL}(T) \\[2mm]
S_{WL} + (S_W - S'_{WL}(T)) \dfrac{S_{OWr} - S_{WL}}{S'_{OWr}(T) - S'_{WL}(T)} & S'_{WL}(T) \leqslant S_W < S'_{OWr}(T) \\[2mm]
\begin{aligned}
S_{OWr} &+ (S_W - S'_{OWr}(T)) \times \\
&\times \dfrac{1 - S_{OWCR} - S_{GL} - S_{OWr}}{1 - S'_{OWCR}(T) - S'_{GL}(T) - S'_{OWr}(T)}
\end{aligned} & S'_{OWr}(T) \leqslant S_W < 1 - S'_{OWCR}(T) - S'_{GL}(T) \\[2mm]
1 - S_{OWCR} - S_{GL} & S_W \geqslant 1 - S'_{OWCR}(T) - S'_{GL}(T)
\end{cases}
$$
$$(3.83)$$

Note that $S'_{OW}(S'_{OWr}(T), T) = S_{OWr}$.

If the keyword KRORW (see 6.6.25) is not specified or $k_{rOW}(S_{OWr}) \geqslant k_{rOmax}$, then $k'_{rOW}(S_W, T)$ is calculated via formula (3.72), else

$$
k'_{rOW}(S_W, T) =
\begin{cases}
\begin{aligned}
k'_{rORW}(T) &+ \dfrac{k'_{rOmax}(T) - k'_{rORW}(T)}{k_{rOmax} - k_{rOW}(S_{OWr})} \times \\
&\times (k_{rOW}(S'_{OW}(S_W, T)) - k_{rOW}(S_{OWr}))
\end{aligned} & S_W < S'_{OWr}(T) \\[2mm]
\dfrac{k'_{rORW}(T)}{k_{rOW}(S_{OWr})} k_{rOW}(S'_{OW}(S_W, T)) & S_W \geqslant S'_{OWr}(T)
\end{cases}
$$
$$(3.84)$$

Note that $k'_{rOW}(S'_{OWr}(T), T) = k'_{rORW}(T)$.

If gas phase is present then for the given S_G, T the following parameters are calculated:

$$
S'_{Gr}(T) =
\begin{cases}
1 - S'_{OGCR}(T) - S'_{WL}(T) & \text{if oil phase is present} \\
1 - S'_{WCR}(T) & \text{else}
\end{cases}
$$

$$
S_{Gr} =
\begin{cases}
1 - S_{OGCR} - S_{WL} & \text{if oil phase is present} \\
1 - S_{WCR} & \text{else}
\end{cases}
$$

If the calculated value $S'_{Gr}(T)$ is not in the interval $[S'_{GCR}(T), S'_{GU}(T)]$ or $S'_{GCR}(T) \geqslant S'_{GU}(T)$, then $S'_G(S_G, T)$ is calculated via formula (3.75), else

$$
S'_G(S_G, T) =
\begin{cases}
S_{GCR} & S_G < S'_{GCR}(T) \\[2mm]
S_{GCR} + (S_G - S'_{GCR}(T)) \dfrac{S_{Gr} - S_{GCR}}{S'_{Gr}(T) - S'_{GCR}(T)} & S'_{GCR}(T) \leqslant S_G < S'_{Gr}(T) \\[2mm]
S_{Gr} + (S_G - S'_{Gr}(T)) \dfrac{S_{GU} - S_{Gr}}{S'_{GU}(T) - S'_{Gr}(T)} & S'_{Gr}(T) \leqslant S_G < S'_{GU}(T) \\[2mm]
S_{GU} & S_G > S'_{GU}(T)
\end{cases}
$$
$$(3.85)$$

Note that $S'_G(S'_{Gr}(T), T) = S_{Gr}$.

If the keyword **KRGR** (see 6.6.27) is not specified or $k_{rG}(S_{Gr}) \geqslant k_{rGmax}$, then $k'_{rG}(S_G, T)$ is calculated via formula (3.76), else

$$
k'_{rG}(S_G, T) =
\begin{cases}
\dfrac{k'_{rGR}(T)}{k_{rG}(S_{Gr})} k_{rG}(S'_G(S_G, T)) & S_G < S'_{Gr}(T) \\[2ex]
k'_{rGR}(T) + \dfrac{k'_{rGmax}(T) - k'_{rGR}(T)}{k_{rGmax} - k_{rG}(S_{Gr})} \times \\[2ex]
\quad \times (k_{rG}(S'_G(S_G, T)) - k_{rG}(S_{Gr})) & S_G \geqslant S'_{Gr}(T)
\end{cases}
\tag{3.86}
$$

Note that $k'_{rG}(S'_{Gr}(T), T) = k'_{rGR}(T)$.

If oil phase is present then $S'_{OG}(S_G, T)$, $k'_{rOG}(S_G, T)$ also are calculated. For the given S_G, T the following parameters are calculated:

$$
S'_{OGr}(T) = S'_{GCR}(T), \quad S_{OGr} = S_{GCR}.
$$

If calculated value $S'_{OGr}(T)$ is not in the interval $[S'_{GL}(T), 1 - S'_{OGCR}(T) - S'_{WL}(T)]$ or $S'_{GL}(T) \geqslant 1 - S'_{OGCR}(T) - S'_{WL}(T)$, then $S'_{OG}(S_G, T)$ is calculated via formula (3.77), else

$$
S'_{OG}(S_G, T) =
$$
$$
=
\begin{cases}
S_{GL} & S_G < S'_{GL}(T) \\[2ex]
S_{GL} + (S_G - S'_{GL}(T)) \dfrac{S_{OGr} - S_{GL}}{S'_{OGr}(T) - S'_{GL}(T)} & S'_{GL}(T) \leqslant S_G < S'_{OGr}(T) \\[2ex]
S_{OGr} + (S_G - S'_{OGr}(T)) \times \\[1ex]
\quad \times \dfrac{1 - S_{OGCR} - S_{WL} - S_{OGr}}{1 - S'_{OGCR}(T) - S'_{WL}(T) - S'_{OGr}(T)} & S'_{OGr}(T) \leqslant S_G < 1 - S'_{OGCR}(T) - S'_{WL}(T) \\[2ex]
1 - S_{OGCR} - S_{WL} & S_G \geqslant 1 - S'_{OGCR}(T) - S'_{WL}(T)
\end{cases}
\tag{3.87}
$$

Note that $S'_{OG}(S'_{OGr}(T), T) = S_{OGr}$.

If the keyword **KRORG** (see 6.6.25) is not specified or $k_{rOG}(S_{OGr}) \geqslant k_{rOmax}$, then $k'_{rOG}(S_G, T)$ is calculated via formula (3.78), else

$$
k'_{rOG}(S_G, T) =
\begin{cases}
k'_{rORG}(T) + \dfrac{k'_{rOmax}(T) - k'_{rORG}(T)}{k_{rOmax} - k_{rOG}(S_{OGr})} \times \\[2ex]
\quad \times (k_{rOG}(S'_{OG}(S_G, T)) - k_{rOG}(S_{OGr})) & S_G < S'_{OGr}(T) \\[2ex]
\dfrac{k'_{rORG}(T)}{k_{rOG}(S_{OGr})} k_{rOG}(S'_{OG}(S_G, T)) & S_G \geqslant S'_{OGr}(T)
\end{cases}
\tag{3.88}
$$

Note that $k'_{rOG}(S'_{OGr}(T), T) = k'_{rORG}(T)$.

3.34.3 Phase relative permeabilities for free-phase systems

If oil phase is present in the model, then oil relative permeability k_{rO} should be calculated. Let's consider models from the table 4.

Name	e300 data format	CMG data format
Linear model	**default**	RPT (see 7.5.2) (default)
Stone I model(standard)	STONE1 (see 6.6.9)	disable
Stone I model(modified)	disable	RPT (see 7.5.2) (STONE1)
Stone II model(modified)	STONE2 (see 6.6.10)	**default**, RPT (see 7.5.2) (STONE2)

Таблица 4: Phase relative permeabilities for free-phase systems

Linear Beyker's model. Let (ε – small parameter):

$$k'_{rO}(S_W, S_G, T) = \begin{cases} k'_{rOG}(S_G + S_W - S'_{WL}(T), T) & S_W - S'_{WL}(T) < \varepsilon \\ \\ k'_{rOW}(S_G + S_W, T) & S_G < \varepsilon \\ \\ \dfrac{S_G \cdot k'_{rOG}(S_G + S_W - S'_{WL}(T), T)}{S_G + (S_W - S'_{WL}(T))} + \\ \quad + \dfrac{(S_W - S'_{WL}(T)) \cdot k'_{rOW}(S_G + S_W, T)}{S_G + (S_W - S'_{WL}(T))} & \text{else} \end{cases} \tag{3.89}$$

First Stone's model. Let's consider $S_{Om}(S_G, T)$:

- for standart first Stone's model:

$$S_{Om}(T) = \min(S'_{OWCR}(T), S'_{OGCR}(T))$$

- for modified first Stone's model:

$$S_{Om}(S_G, T) = \alpha(S_G, T)S'_{OWCR}(T) + (1 - \alpha(S_G, T))S'_{OGCR}(T),$$

где

$$\alpha(S_G, T) = 1 - \frac{S_G}{1 - S'_{WL}(T) - S'_{OGCR}(T)}.$$

Let

$$S'_W(S_W, T) = \max(S_W, S'_{WL}(T)).$$

Let's denote as $S^*_O(S_W, S_G, T)$, $S^*_W(S_W, S_G, T)$, $S^*_G(S_G, T)$ the following values:

$$S^*_O(S_W, S_G, T) = \begin{cases} \dfrac{1 - S'_W(S_W, T) - S_G - S_{Om}(S_G, T)}{1 - S'_{WL}(T) - S_{Om}(S_G, T)} & \text{if } 1 - S'_W(S_W, T) - S_G \geqslant S_{Om}(S_G, T) \\ 0 & \text{else} \end{cases}$$

$$S^*_W(S_W, S_G, T) = \frac{S'_W(S_W, T) - S'_{WL}(T)}{1 - S'_{WL}(T) - S_{Om}(S_G, T)}$$

$$S^*_G(S_G, T) = \frac{S_G}{1 - S'_{WL}(T) - S_{Om}(S_G, T)}$$

Then

$$k'_{rO}(S_W, S_G, T) = k'_{rOW}(S'_{WL}(T), T) \cdot \frac{k'_{rOW}(S_W, T)}{k'_{rOW}(S'_{WL}(T), T)} \cdot \frac{k'_{rOG}(S_G, T)}{k'_{rOG}(S'_{GL}(T), T)}$$
$$\cdot \frac{S^*_O(S_W, S_G, T)}{(1 - S^*_W(S_W, S_G, T))(1 - S^*_G(S_G, T))} \quad (3.90)$$

Since $k_{rOW}(S_{WL}, T) = k_{rOG}(S_{GL}, T)$ – maximal value in the table, this formula provides $k_{rO}(S_{WL}, S_G, T) = k_{rOG}(S_G, T)$ and $k_{rO}(S_W, S_{GL}, T) = k_{rOW}(S_W, T)$.

Second Stone's model.   Let's consider the following functions:

$$\alpha(T) = \begin{cases} \dfrac{k'_{rGmax}(T)}{k'_{rWmax}(T)} & \text{if KRG (see 6.6.27) and KRW (see 6.6.26) are specified} \\[2mm] \dfrac{k_{rWmax}}{k'_{rWmax}(T)} & \text{if only KRW (see 6.6.26) is specified} \\[2mm] \dfrac{k'_{rGmax}(T)}{k_{rWmax}} & \text{if only KRG (see 6.6.27) is specified} \\[2mm] 1 & \text{else} \end{cases} \quad (3.91)$$

$$\beta(T) = \begin{cases} \dfrac{k'_{rWmax}(T)}{k'_{rGmax}(T)} & \text{if KRG (see 6.6.27) and KRW (see 6.6.26) are specified} \\[2mm] \dfrac{k'_{rWmax}(T)}{k_{rGmax}} & \text{if only KRW (see 6.6.26) is specified} \\[2mm] \dfrac{k_{rGmax}}{k'_{rGmax}(T)} & \text{if only KRG (see 6.6.27) is specified} \\[2mm] 1 & \text{else} \end{cases} \quad (3.92)$$

Then

$$k'_{rO}(S_W, S_G, T) = k'_{rOW}(S'_{WL}(T), T) \left(\left(\frac{k'_{rOW}(S_W, T)}{k'_{rOW}(S'_{WL}(T), T)} + \alpha(T) k'_{rW}(S_W, T) \right) \right.$$
$$\left. \cdot \left(\frac{k'_{rOG}(S_G, T)}{k'_{rOG}(S'_{GL}(T), T)} + \beta(T) k'_{rG}(S_G, T) \right) - (\alpha(T) k'_{rW}(S_W, T) + \beta(T) k'_{rG}(S_G, T)) \right) \quad (3.93)$$

If $k_{rO}(S_W, S_G, T) < 0$, then $k_{rO}(S_W, S_G, T) = 0$ is considered. Since $k_{rOW}(S_{WL}, T) = k_{rOG}(S_{GL}, T)$ – maximal value in the table, this formula provides $k_{rO}(S_{WL}, S_G, T) = k_{rOG}(S_G, T)$ and $k_{rO}(S_W, S_{GL}, T) = k_{rOW}(S_W, T)$.

4 Units

A table of units supported:

Quantity	SI	METRIC	FIELD	LAB
Length, depth, radius	m	m	ft	cm
Time	s	day	day	hr
Density	kg/m^3	kg/m^3	lbl/ft^3	kg/m^3
Pressure (absolute)	Pa	Barsa	Psia	Atma
Pressure (difference)	Pa	Bars	Psi	Atm
Temperature (absolute)	°K	°K	°R	°K
Temperature (difference)	°K	°K	°F	°K
Compressibility	1/Pa	1/Bars	1/Psi	1/Atm
Viscosity	Pa*s	cpoise	cpoise	cpoise
Permeability	m^2	mDarcy	mDarcy	mDarcy
Liquid surface volume	m^3	sm^3	stb	scc
Gas surface volume	m^3	sm^3	Mscf	scc
Reservoir volume	m^3	rm^3	RB	rcc
Liquid surface volume rate	m^3/s	sm^3/day	stb/day	scc/hr
Gas surface volume rate	m^3/s	sm^3/day	Mscf/day	scc/hr
Reservoir volume rate	m^3/s	rm^3/day	RB/day	rcc/hr
Rate per unit area	m^3/s/m^2	m^3/day/m^2	stb/s/ft^2	scc/hr/cm^2
Formation volume factor (liquid)	m^3/m^3	rm^3/sm^3	RB/stb	rcc/scc
Formation volume factor (gas)	m^3/m^3	rm^3/sm^3	RB/Mscf	rcc/scc
Gas-oil ratio	m^3/m^3	sm^3/sm^3	Mscf/stb	scc/scc
Oil-gas ratio	m^3/m^3	sm^3/sm^3	stb/Mscf	scc/scc
Transmissibility	$\dfrac{m^3}{s*Pa}$	$\dfrac{cp*m^3}{day*Bars}$	$\dfrac{cp*RB}{day*Psi}$	$\dfrac{cp*cm^3}{hr*Atm}$

Unit conversion factor β_c depends on the unit system and defined as follows:

- $\beta_c = 0.00852702$ for metric unit system;

- $\beta_c = 0.00112712$ for field unit system.

Gravity conversion factor g depends on the unit system and is equal to:

- $g = 0.0000980665$ for metric unit system;

- $g = 0.00694$ for field unit system,

UNITS (see 6.1.10).

Standard conditions pressure and temperature in metric system (METRIC) are:

- $p = 1.01325$ bars;

- $T = 16°C$

In case of compositional run, those conditions can be changed by user, STCOND (see 6.7.7).

If units are not specified explicitly, then METRIC system is used by default. In the current version the only systems METRIC and FIELD are supported.

5 Mathematical model

Simulator uses usual finite difference approximation with respect to space and time variables to obtain discretization of physical model equations.

5.1 Space approximation

Simulator uses standard finite difference approximation with respect to space variables on rectangular block centered mesh. The standard upstream approximation is used for computation of coefficients of equations depending on saturations.

5.2 Time approximation

For approximation with respect to time simulator uses Fully Implicit method. In this case both pressures and component molar densities are regarded as unknowns and all coefficients are calculated from their current values. This leads to system of non-linear equations to be solved. For solution standard Newton method is used.

Time step is chosen automatically to provide method convergence.

5.3 Well Approximation

After discretization of equations for each grid block penetrated by the well the relationship between the flow rate of each phase, pressure in the well bore and grid block pressure should be established. This relationship is called "Inflow performance relationship". This relationship is used to calculate bottom hole pressure if user specifies rate control for injection or production well or to calculate rate if user specifies bottom hole pressure control for a well (WCONPROD (see 6.10.12), WCONINJE (see 6.10.14)).

5.3.1 Well Inflow Performance

The term "connection"denotes the flow path between the well bore and a single reservoir grid block. While calculating inflow performance relationship for each connection the following assumptions are made:

- The well is assumed to penetrate the full thickness of the block, through its center, perpendicularly to two of its faces.

- For any calculation time step density of fluid within the well bore does not vary with depth.

- Friction effects in the well bore are neglected.

- Capillary pressure is neglected when calculating inflow performance relationship; oil phase pressure is used.

After discretization of equation (2.62) we get the following inflow performance relationship for each well connection l with coordinates (i, j, k), written in terms of the volumetric phase flow rate ($P = \{W, O, G\}$, i.e. Water, Oil, Gas):

$$Q_P^l(p^l, \mathbf{N}^l, t) = T^l(t) \cdot M_P(p^l, S_W^l, S_G^l)(p^l - p_{con}^l(t)) \tag{5.1}$$

where

- $Q_P^l = Q_P^l(p^l, \mathbf{N}^l, t)$ – volumetric flow rate of phase P through connection l in reservoir conditions;

- $T^l(t)$ – connection productivity index, defined below, see section 5.3.2;

- $M_P = M_P(p^l, S_W^l, S_G^l)$ – the total phase mobility at connection l, defined below, see section 5.3.5;

- p^l, S_W^l, S_G^l – nodal pressure and saturations in the grid block, containing connection;

- N_c – component molar densities in block l;

- $p_{con}^l(t)$ – the connection l pressure, see section 5.3.6;

5.3.2 Connection transmissibility calculation

Connection productivity index $T^l(t)$ may be defined by user, COMPDAT (see 6.10.6). Otherwise it is calculated according to formula

$$T^l(t) = \frac{2\pi K_{mult}^l(t)\beta_c (Kh)^l}{(\log(r_o^l/r_w^l) + s^l)}. \tag{5.2}$$

Here

- $K_{mult}^l(t)$ – KH multiplier for connection l, COMPDAT (see 6.10.6));

- β_c – the unit conversion factor, see section 4;

- $(Kh)^l$ may be explicitly defined by user, COMPDAT (see 6.10.6); otherwise it is calculated as product of K^l – average permeability in plane perpendicular to well axis, see section 5.3.3, and h^l, the size of the grid block in the direction of perforation penetration;

- r_o^l – pressure equivalent radius, defined below, see section 5.3.4;

- $r_w^l = d_w^l/2$ – the radius of the well bore at the connection l;

- s^l – the skin factor at the connection l.

5.3.3 Average permeability calculation

Average permeability K^l for diagonal permeability tensor is calculated as geometric average of two orthogonal component of tensor. That is,

- for Z-directed well K^l could be calculated as $K^l = (k_x^l k_y^l)^{1/2}$

- for X-directed well K^l could be calculated as $K^l = (k_y^l k_z^l)^{1/2}$

- for Y-directed well K^l could be calculated as $K^l = (k_x^l k_z^l)^{1/2}$

Here k_x^l, k_y^l, k_z^l are elements of permeability tensor corresponding to block l.

5.3.4 Pressure equivalent radius calculation

The pressure equivalent radius r_o^l is defined as the distance from the well at which the pressure, calculated using (5.1) is equal to the pressure of the block, containing the connection. In a Cartesian grid the Peaceman's formula is used, which is applicable to rectangular grid blocks in case of permeability anisotropy. As mentioned above, the well is assumed to penetrate the full thickness of the block, through its center, perpendicularly to two of its faces.

The pressure equivalent radius for the connection l is calculated as follows:

$$r_o^l = 0.28 \frac{\left(D_1^{l\,2} \cdot \left(\frac{k_2^l}{k_1^l} \right)^{1/2} + D_2^{l\,2} \cdot \left(\frac{k_1^l}{k_2^l} \right)^{1/2} \right)^{1/2}}{\left(\frac{k_2^l}{k_1^l} \right)^{1/4} + \left(\frac{k_1^l}{k_2^l} \right)^{1/4}} \tag{5.3}$$

where

- D_1^l and D_2^l are the sizes of the grid block, containing connection, in the dimensions perpendicular to well penetration;

- k_1^l and k_2^l are the permeabilities of the grid block, containing connection, in the dimensions perpendicular to well penetration.

5.3.5 Mobility calculation

Mobility calculations are different for production well connections and injection well connections. For production well connections, the mobility depends on the conditions in the grid block containing the connection. For injection well connections we use downstream approximation as a standard practice to calculate phase mobility. It results in injection phase mobility being equal to sum of all three phases mobilities.

For production wells phase mobility is calculated as:

$$M_P(p^l, S_W^l, S_G^l) = \frac{k_{rP}(S_W^l, S_G^l)}{\mu_P(p^l)} \tag{5.4}$$

For injection wells phase mobility is calculated as:

$$M_P(p^l, S_W^l, S_G^l) = \begin{cases} \dfrac{k_{rO}(S_W^l)}{\mu_O(p^l)} + \dfrac{k_{rW}(S_W^l, S_G^l)}{\mu_W(p^l)} + \dfrac{k_{rG}(S_G^l)}{\mu_G(p^l)}, & \text{for injected phase} \\ 0, & \text{for other two phases} \end{cases} \tag{5.5}$$

where

- $k_{rP} = k_{rP}(S_W^l, S_G^l)$ is phase relative permeability evaluated at grid block saturations;

- $\mu_P = \mu_P(p^l)$ is phase viscosity evaluated at grid block conditions;

5.3.6 Average well bore density and connection pressure calculation

Friction effects are usually small within the well bore at formation level, and they are neglected. We assume Darcy flow. The average well bore density is assumed to be constant at each calculation time step and equal to:

$$\bar{\rho}_{av} = \frac{\sum\limits_{l} \rho_{O,SC}\tilde{q}_O^l + \rho_{W,SC}\tilde{q}_W^l + \rho_{G,SC}(\tilde{q}_G^l + R_{G,O}(p^l)\tilde{q}_O^l)}{\sum\limits_{l} B_O(p_{av})\tilde{q}_O^l + B_w(p_{av})\tilde{q}_W^l + B_G(p_{av})(\tilde{q}_G^l + (R_{G,O}(p^l) - R_{G,O}(p_{av}))\tilde{q}_O^l)} \tag{5.6}$$

where

- $\rho_{P,SC}$ is the density of phase P at standard conditions;

- $B_P = B_P(p_{av})$ is the phase formation volume factor evaluated at well bore conditions;

- p_{av} is average well bore pressure;

- $\tilde{q}_P^l = \tilde{q}_P^l(p^l, S_W^l, S_G^l, t)$ is the well phase volumetric flow rate into connection l at standard conditions;

In this case connection pressure $p_{con}^l(t)$ is calculated as

$$p_{con}^l(t) = p_{BH}(t) - \bar{\rho}_{av}(t)g(D^l - D_{BH}) \tag{5.7}$$

here

- $p_{BH}(t)$ is bottom hole pressure of the well, user specified or calculated from (2.62), WCONPROD (see 6.10.12), WCONINJE (see 6.10.14);

- D_{BH} is bottom hole depth;

- g is gravity constant;

- D^l is depth of connection l,

- average wellbore density $\bar{\rho}_{av}(t)$ is calculated according to (5.6).

5.4 Modified well model

Let's consider modified well model. Modifications help to describe behind-the-casing flow and hydraulic fracture simulation.

5.4.1 Well model with generalized connections

For each connection located in grid block l, the following data is specified:

- Numbers of grid blocks from which the connection produces (or injects):

$$l_0, l_1, \ldots, l_{m_l}$$

where $l_0 = l$.

- Resistance (connection effective multipliers) between the connection and corresponding blocks (from which this connection produces or injects):

$$\gamma_0^l, \gamma_1^l, \ldots, \gamma_{m_l}^l$$

where $\gamma_0 = 1$.

- Threshold pressures. If these values are exceeded the flow from corresponding blocks to connection starts (or the flow from connection into corresponding grid blocks):

$$p_0^l, p_1^l, \ldots, p_{m_l}^l$$

where $p_0 = 0$.

For each grid block which is connected to the well connection we assume:

- Pressure in connection is calculating via the same formulae

- Connection productivity index between blocks $l = l_0$ and l_i is calculated via formula without the skin factor but with the multiplier γ_i^l:

$$\widehat{\Theta}^{w,l_i} = \gamma_i^l \Upsilon \frac{2\pi K_{mult}^{w,l_i} \beta_c K^{w,l_i} h^{w,l_i}}{\log(r_0^{w,l_i}/r_w^{w,l_i})} = \gamma_i^l \Theta^{w,l_i}, \tag{5.8}$$

- The inflow perfomance relationship for the well connection l from the grid block $l_{l,i}$ (when $p^{l_i} > p^{w,l}$) is specified with the replacement of the multiplier $(p^{l_i} - p^{w,l})$ (if $l_i \neq 0$) by $\Psi(p^{l_i} - p^{w,l}, p_{l_i}^l)$, where

$$\Psi(x,y) = \begin{cases} x - y & \text{if } x > y \\ x + y & \text{if } x < -y \\ 0 & \text{else} \end{cases}$$

- The injection from the connection l to the block $l_{l,i}$ (when $p^{l_i} < p^{w,l}$) is specified with the replacement of multiplier $(p^{l_i} - p^{w,l})$ (if $l_i \neq 0$) by $\Psi(p^{l_i} - p^{w,l}, p^l_{l_i})$.

Thus, the total inflow (outflow) of the component c in (from) the connection l is:

$$
q_c^l = -\sum_{i\,:\,p^{l_i} > p^{w,l}} \gamma_i^l\, \Theta^{w,l_i}\, \Psi(p^{l_i} - p^{w,l}, p^l_{l_i}) \left(\sum_{P=1}^{n'_P} x_{c,P}(p^{l_i}, N^{l_i}, T^{l_i})\, \xi_P(p^{l_i}, N^{l_i}, T^{l_i})\, M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right)
$$

$$
- \frac{q_c^w}{q_{tot}^w} \sum_{i\,:\,p^{l_i} < p^{w,l}} \gamma_i^l\, \Theta^{w,l_i}\, \Psi(p^{l_i} - p^{w,l}, p^l_{l_i})\, \xi_{avg}(p^l, \boldsymbol{q}^w, T^l) \left(\sum_{P=1}^{n'_P} M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right)
$$

5.4.2 Modified well model and behind-the-casing flow

To allow the calculation of behind-the-casing flow in the hydrodynamic model should be added well trajectories data. This data is used to make the list of grid blocks, which contain the well bore (status PATH in the keyword COMPDAT (see 6.10.6); manually input of status PATH in graphical interface). Hydrodynamic model in Eclipse data format is not enough to calculate behind-the-casing flow because Eclipse keywords provide only input of perforated intervals (not all well bore trajectory).

1. To input *the well bore trajetory* there are the following ways:

    - **Manually input** of well bore trajetory in graphical interface.
    - **Automatic calculation** of well bore trajetory for vertical wells:
        - consecutive well connection (situated in different Z layers) are connected in well trajectory;
        - trajectory is constructed from the last well connection (the deepest one) to the bottom layer of reservoir.
    - **Load file** with well trajectory.

2. To input *connection links because of behind-the-casing flow* there are the following ways:

    - **Manually input** of additional links among several well connections.
    - **Linking of all well connections**.

3. To input *connection effective multipliers* between the connections because of behind-the-casing flow (if the option **Behind-the-casing flow** is enable) there are the following ways:

    - **Manually input** of connection effective multipliers (the keyword WNNC (see 6.10.39)).
    - **Automatic calculation** of connection effective multipliers using the data of the keyword WCONNNC (see 6.10.40) (see the formulae below).

Let's consider method of **Automatic calculation** of connection effective multipliers. Let for every connection l, $l = 1, \dots, L$ **and well trajectory** are specified:

- effective permeability along the well bore K^l (it is equal to the product of fracture permeability and the square of it's cross-section);

- permeability multiplier along the well bore M^l, which allows to specify individual properties of connection l, whithout influence on other zones;

- threshold pressure p^l.

Well connections and trajectory parts are numbered along the well trajectory:

$$\boxed{K_1}\ \boxed{\ }\ \boxed{\ }\ \boxed{\ }\ \boxed{K_l}\ \boxed{\ }\ \boxed{\ }\ \boxed{K_L}$$

For the connection l, linked with the connection or trajectory part i (index j includes only the numbers of connections and trajectory parts which are linked with l):

$$\gamma_i^l = \frac{M_l}{\sum\limits_{j=i}^{l}\frac{1}{K^j}} \quad \text{if } i < l, \qquad \gamma_i^l = \frac{M_l}{\sum\limits_{j=l}^{i}\frac{1}{K^j}} \quad \text{if } i > l. \tag{5.9}$$

These formulae are analogues to the calculation of grid blocks transmissibility coefficients and provide:

- **zero value** $\gamma_i^l = 0$, if effective permeability K^j **of at least one** connection or trajectory part between l and i is zero;

- **accumulated resistance** to flow between connections l and i because of **length and permeability** of path between them.

Thus, the total inflow (outflow) of the component c in (from) connection l is:

$$q_c^l = -\sum_{i \neq l : p^{l_i} > p^{w,l}} \gamma_i^l \, \Theta^{w,l_i} \, \Psi(p^{l_i} - p^{w,l}, p^l) \left(\sum_{P=1}^{n_P'} x_{c,P}(p^{l_i}, N^{l_i}, T^{l_i}) \, \xi_P(p^{l_i}, N^{l_i}, T^{l_i}) \, M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right)$$

$$- \frac{q_c^w}{q_{tot}^w} \sum_{i \neq l : p^{l_i} < p^{w,l}} \gamma_i^l \, \Theta^{w,l_i} \, \Psi(p^{l_i} - p^{w,l}, p^l) \, \xi_{avg}(p^l, \boldsymbol{q}^w, T^l) \left(\sum_{P=1}^{n_P'} M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right) + \widehat{q}_c^l,$$

where $\widehat{q}_c^l$ – inflow (outflow) when behind-the-casing flow is disable.

If there is not enough information all values M^l, p^l can be assumed to be equal for all well connections and trajectory parts.

5.4.3 Modified well model and hydraulic fracture simulation

For the hydraulic fracture simulation the following parameters should be specified:

- well name and date;

- proppant properties (dependence between proppant permeability and the pressure) (number of proppants – NPROPANTS (see 6.5.16), proppant names – PROPANTNAMES (see 6.5.17), the table of relation between pressure and proppant permeability – PROPANTTABLE (see 6.5.18));

- azimuth angle of hydraulic fracture;

- half of fracture lenght;

- fracture width;

- fracture height (numbers of first and last connection);

- zenithal angle.

In mathematical model we should describe:

- links between connections and grid blocks (because of hydraulic fracture);

- flow along hydraulic fracture;

- total inflow to hydraulic fracture.

Hydraulic fracture could be specified via the keywords WFRAC (see 6.10.41), WFRACP (see 6.10.43), COMPFRAC (see 6.10.45).

5.4.4 Hydraulic fractures

To calculate the flow in the reservoir after the hydraulic fracture the trajectory of hydraulic fracture should be specified. There are the following ways:

- **Manually input** of additional links between several well connections and grid blocks (because of hydraulic fracture).

- **Automatic creation of links** between the given connection and all grid blocks in the parallelepiped (specified by azimuth angle, half of fracture lenght, fracture width and height).

5.4.5   Flow rate along the fracture

Let for every connection l, linked (because of hydraulic fracture) with block i (index j include only blocks with numbers between blocks l and i):

$$\gamma_i^l = \frac{M_l}{\sum\limits_{j=i}^{l} \frac{1}{K^j}} \tag{5.10}$$

where K^j – permeability along the fracture in block j. These formulae are analogues to the calculation of grid blocks transmissibility coefficient and provide:

- **zero value** $\gamma_i^l = 0$, if permeability K^j **of at least one** fracture part between parts l and i is zero;

- **accumulated resistance** to flow between l and i because of **lenght and permeability** of path between them.

To simulate the dependence between hydraulic fracture and permeability, proppant washout, fraction plugging and pressure – K^j can depend on:

- fracture area and proppant properties;

- pressure in the block, through which the fraction goes;

- phase flow through blocks with fracture;

- time.

5.4.6   Calculation of the inflow to the fracture from the grid block

Let us consider the following scheme. The fracture in the block j is replaced by the virtual connection with the following conditions:

- perimeter of well bore cross-section is equal to fracture perimeter (that provides the formula to calculate well diameter);

- lenght of perforated interval is equal to the well bore lenght in the block;

- the inflow to the well connection is calculated using block permeability in the direction, which is orthogonal to fracture border with maximal area.

Via this data we can calculate permeability multiplier of this virtual connection ($\tilde{\Theta}^{w,j}$), and then calculate the inflow via common formula.

On the picture 1 there is a fracture which goes along Y-axis, height h and width w (less then h). Inflow to this fraction is equal to the inflow to the well with diameter $d = 2(w+h)/\pi$,
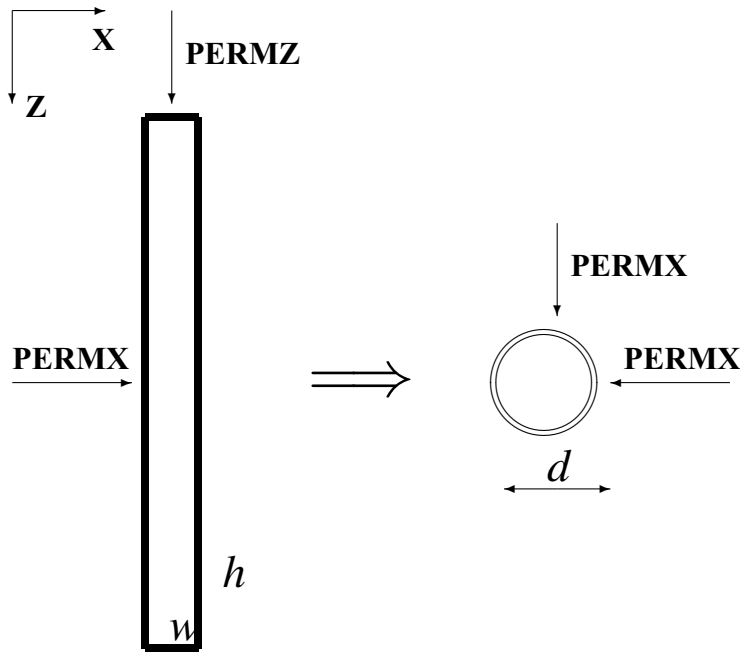
Рис. 1: Calculation of equivalent inflow to the fracture

Y-orientation and which is situated in the grid block. X and Z permeability of this block is equal to X permeability of the block with fracture. So we obtain

$$\tilde{\Theta}^{w,j} = \frac{2\pi\beta_c K^{w,j} h^{w,j}}{\log(r_0^{w,j}/r_w^{w,j})}, \quad r_w^{w,j} = (w+h)/\pi,$$

$$r_0^{w,j} = 0.14 * \sqrt{\mathbf{DX}^2 + \mathbf{DZ}^2}, \quad K^{w,j} = \mathbf{PERMX}, \quad h^{w,j} = \mathbf{DY}$$

where $\mathbf{DX}$, $\mathbf{DY}$, $\mathbf{DZ}$ – geometric sizes of block through which the fraction goes.

5.4.7 Total inflow from the hydraulic fraction to the well connection

l_i stands for numbers of grid blocks which are connected to the well connection l. Then total inflow (outflow) of component c to the well connection l from hydraulic fracture:

$$\tilde{q}_c^l = - \sum_{i \neq l : p^{l_i} > p^{w,l}} \gamma_i^l \tilde{\Theta}^{w,l_i} \Psi(p^{l_i} - p^{w,l}, p^l) \left(\sum_{P=1}^{n_P'} x_{c,P}(p^{l_i}, \mathbf{N}^{l_i}, T^{l_i}) \xi_P(p^{l_i}, \mathbf{N}^{l_i}, T^{l_i}) M_P(p^{l_i}, \mathbf{N}^{l_i}, T^{l_i}) \right)$$

$$- \frac{q_c^w}{q_{tot}^w} \sum_{i \neq l : p^{l_i} < p^{w,l}} \gamma_i^l \tilde{\Theta}^{w,l_i} \Psi(p^{l_i} - p^{w,l}, p^l) \xi_{avg}(p^{l_i}, \mathbf{q}^w, T^{l_i}) \left(\sum_{P=1}^{n_P'} M_P(p^{l_i}, \mathbf{N}^{l_i}, T^{l_i}) \right)$$

This value should be added to the common inflow to the connection (or to the inflow which is calculated using behind-the-casing flow).

5.4.8 Description of simulation of large amount of hydraulic fractures

If fraction height and lenght are big numbers (see the description of fraction in the section 5.4.3) then the model of fraction as linked blocks is not effective because of complicated calculations. So we consider the hydraulic fracture the following way: there is a virtual connection in each block through which the fraction goes. For this virtual connection we describe inflow formula and formula of it's transportation to the well bore. An advantage of this method is simplification of calculations, and a disadvantage – it's impossible to divide an inflow to the fraction among well conections (i.e. inflow is calculated as inflow from fraction to well, not to specific well connection).

The fraction is specified according to the sections 5.4.3, 5.4.4. $l_1, \ldots, l_L$ stand for the blocks through which the fraction goes, average fracture perimeter in the block l_i – Π_{l_i}, average fracture lenght in the block l_i – L_{l_i}, area of fracture cross-section in the block l_i – S_{l_i}, the distance from block l_i to the well bore – D_{l_i}. For simplification of designation let consider fracture plane as vertical plane and quasiorthogonal to OX, see the picture pic. 1. To other location of fraction calculations are similar. To calculate iflow from block l_i to the fracture we use the formula from the section 5.4.6:

$$\tilde{\Theta}^{w,l_i} = \frac{2\pi\beta_c \cdot \mathbf{PERMX} \cdot L_{l_i}}{\log\left(0.28 * \pi * \sqrt{\mathbf{DX}^2 + \mathbf{DZ}^2}/S_{l_i}\right)},$$

To calculate flow along the fracture we should approximate an expression (5.10 efficiency of flow along fracture) (in fact it is the multiplier to $\tilde{\Theta}^{w,l_i}$). Proppant properties for the fracture should be specified:

- **proppant permeability** (mD, using PROPANTTABLE (see 6.5.18)) $K(p)$ is the function of pressure in the block; if this function is not specified then $K = \infty$;

- **dependence between permeability and flow** of phase or liquid; This dependence is specified flow function f of dimensionless flow s (total phase flow divided by total pore volume of this block):

  1. $f(0) = 1$;
  2. $f(s) \geq 0$ for all s.

There are the following types of this function:

1. Exponential: $f(s) = k + (1-k) * \exp(-a*s)$, Parameters: $k \geq 0$, $a > 0$ are specified via FLOWFUNC (see 6.5.19) (function type EXP).

2. Linear: $f(s) = \max\{1 + (k-1) * a * s, 0\}$, Parameters: $k \geq 0$, $a > 0$ are specified via FLOWFUNC (see 6.5.19) (function type LIN).

3. Tabular: the table $(s, f(s))$ is specified via keyword FLOWFTAB (see 6.5.22) (the number of tabular functions is specified via the keyword NFLOWFTB (see 6.5.20), tabular function names – FLOWFNAMES (see 6.5.21)).

If function is not specified, then $f(s) = 1$.

Resultant effective permeability of proppant in the block l_i with the area of cross-section S_{l_i} is equal to

$$K_{l_i}(p^{l_i}, s) = K(p^{l_i}) \cdot S_{l_i} \cdot f(s).$$

Function $K_{l_i}(p^{l_i}, s)$ is dimensionless. Flow efficiency along the fracture from block l_i to the well γ_{l_i} (see (5.10)) is such that:

1. γ_{l_i} is the function only of D_{l_i} if $K(p) = \infty$, $\gamma_{l_i} = 1$ if $K(p) = \infty$ and $D_{l_i} = 0$, this corresponds to block drainage (the well goes through this block);

2. $\gamma_{l_i} = 0$ if $K(p) = 0$;

3. $\gamma_{l_i} = 0$ if $s = \infty$.

Let's consider the following function:

$$\gamma_{l_i}(p^{l_i}, s) = \frac{K_{l_i}(p, s)}{1 + K_{l_i}(p, s)} \cdot \frac{1}{1 + D_{l_i}/L_{l_i}}. \tag{5.11}$$

$\gamma_{l_i}(p^{l_i}, s)$ is dimensionless.

Since the rate of virtual perforations doesn't correspond to specific connection, then we take bottom hole pressure p^w for the pressure in the well and use the function Ψ with threshold value $\widehat{p}^w$, which is the same for all fraction blocks (default value is 0). So we have:

$$\tilde{q}_c^l = - \sum_{i \neq l\,:\,p^{l_i} > p^w} \gamma_{l_i} \tilde{\Theta}^{w,l_i} \Psi(p^{l_i} - p^w, \widehat{p}^w) \left(\sum_{P=1}^{n'_P} x_{c,P}(p^{l_i}, N^{l_i}, T^{l_i}) \xi_P(p^{l_i}, N^{l_i}, T^{l_i}) M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right)$$

$$- \frac{q_c^w}{q_{tot}^w} \sum_{i \neq l\,:\,p^{l_i} < p^w} \gamma_{l_i} \tilde{\Theta}^{w,l_i} \Psi(p^{l_i} - p^w, \widehat{p}^w) \xi_{avg}(p^{l_i}, q^w, T^{l_i}) \left(\sum_{P=1}^{n'_P} M_P(p^{l_i}, N^{l_i}, T^{l_i}) \right)$$

This value should be added to the common inflow to the well (or to the inflow which is calculated using behind-the-casing flow).

To calculate $\gamma_{l_i} = \gamma_{l_i}(p^{l_i}, s)$ we take total flow s and pressure p^{l_i} in block l_i at previous time step (to minimize additional nonlinearity). Pore volume of block (during the calculation of dimensionless flow s) is calculated once when hydraulic fracture is created.

5.4.9 Simulation of plugging of well bottom zone

To simulate plugging of well bottom zone the effectivity multiplier $K_{mult}^{w,l}$ is usually used – WPIMULT (see 6.10.10). We add the function which depends of dimensionless flow s (see the description in the section 5.4.8):

$$\Theta^{w,l} = \Upsilon \frac{2\pi f_p^{w,l}(s) K_{mult}^{w,l} \beta_c K^{w,l} h^{w,l}}{\log(r_0^{w,l}/r_w^{w,l}) + s^{w,l}}, \tag{5.12}$$

where the type of $f_p^{w,l}(s)$ is specified via FLOWFUNC (see 6.5.19) (function type EXP or LIN), FLOWFTAB (see 6.5.22), and for the well connection it is specified via WPIFUNC (see 6.10.48), which is analogous to WPIMULT (see 6.10.10).

After an operation of cleaning of well bottom zone one should specify WPIMULT (see 6.10.10) again, this will set to zero value accumulated dimensionless flow. Specification of other parameters of the function $f_p^{w,l}(s)$ or specification of new type of function will also set to zero accumulated dimensionless flow.

In classical way of hydraulic fracture simulation via skin-factor $s^{w,l}$ in (5.12) one can also specify it's dependence on dimensionless flow s (see the section 5.4.8):

$$\Theta^{w,l} = \Upsilon \frac{2\pi f_p^{w,l}(s) K_{mult}^{w,l} \beta_c K^{w,l} h^{w,l}}{\log(r_0^{w,l}/r_w^{w,l}) + f_s^{w,l}(s) \cdot s^{w,l}}, \tag{5.13}$$

where $f_s^{w,l}(s)$ type is specified via FLOWFUNC (see 6.5.19) (function type EXP or LIN), FLOWFTAB (see 6.5.22), and for the well connection is specified via WSKFUNC (see 6.10.49), analogous to WPIFUNC (see 6.10.48).

After an operation of cleaning of well bottom zone or creation a new fracture one should specify the value of skin-factor $s^{w,l}$ in COMPDAT (see 6.10.6) again. This will set to zero value accumulated dimensionless flow. Specification of other parameters of the function $f_s^{w,l}(s)$ or specification of new type of function will also set to zero accumulated dimensionless flow.

5.4.10 Simulation of well bottom zone dynamics: processing acids, surfactants

After processing acids, surfactants etc. around the well there will be a zone with radius r_{bhz} and permeability K_{bhz}. Modified formula of inflow (5.13), included this zone, will be the following:

$$\Theta^{w,l} = \Upsilon \left(2\pi f_p^{w,l}(s) K_{mult}^{w,l} \beta_c h^{w,l} \right) \frac{K^{w,l} K_{bhz}^{w,l}}{K^{w,l} \log(r_{bhz}^{w,l}/r_w^{w,l}) + K_{bhz} \log(r_0^{w,l}/r_{bhz}^{w,l}) + f_s^{w,l}(s) \cdot s^{w,l}}, \tag{5.14}$$

where parameters $r_{bhz}^{w,l}$, $K_{bhz}^{w,l}$ for the well connection are set via WBHZONE (see 6.10.50). Vertical effective permeability $K^{w,l}$ can also be specified there (formula (5.9 for behind-the-casing flow).

5.5 Solution algorithm for time step problem

Simulator has to solve system of linear equations for pressure on each time step. The preconditioned biconjugate gradient method is used with modified incomplete LU factorization (MILU(0)) as preconditioner. Time step is chosen to satisfy constraints posed on maximal variation of saturation, pressure and pore volume in each grid block.

6   Keywords compatible with tNavigator and Eclipse

For best adaptation of user experience the keyword notations are chosen to be close as much as possible to the most common simulators:

- Eclipse (c) Schlumberger,

- IMEX (c) Computer Modelling Group Ltd,

- STARS (c) Computer Modelling Group Ltd,

- MORE (c) Roxar.

Eclipse compatible keywords are red. For example: TABDIMS (see 6.1.13). Index of Eclipse compatible keywords – 11.

CMG compatible keywords are pink. For example: TEMR (see 8.4.4). Index of CMG compatible keywords – 12.

MORE compatible keywords are green. For example: IDATe (see 9.1.5). Index of MORE compatible keywords – 14.

The keywords which RFD uses only in tNavigator are blue. For example: REACCONC (see 6.8.46). Index of RFD compatible keywords – 13.

In the description of each keyword there is a table where a simulator is ticked off if this keyword is compatible with this simulator.

tNavigator reads keyword notations of these simulators and converts them into its inner data notations.
This section describes all keywords which can be used in tNavigator:

- tNavigator keywords;

- Eclipse 100 keywords;

- Eclipse 300 keywords;

This description pointed out if there are parameters of the keyword which are ignored by tNavigator or which use is different from Eclipse.

In tNavigator there are no strictly defined sections and their order is not fixed. For convenience keyword descriptions are groupped in several sections.

- Definition section (6.1)

- Grid section (6.2)

- Arithmetic section (6.3)

- Region section (6.4)

- Property section (6.5)

- Relative permeabilities and capillary pressures (6.6)

- Compositional properties (6.7)

- Thermal properties (6.8)

- Initialization section (6.9)

- Schedule section (6.10)

- Inflow from aquifer (6.11)

The keyword should be specified in its section. The order in which keywords are listed in the section does not matter, except for the schedule section (6.10) that should conclude the user input data.

All keyword names are case insensitive. Values of parameters usually follow on the next string. If keyword value is not specified by user the simulator uses a default value if it is defined or an error message is issued.

Data tables are terminated by / symbol. If less elements than array length are specified than the rest elements are filled by default values if possible, otherwise an error message is issued. For equal values the standard form NUM*VAL can be used, where NUM – is the number of equal values, and VAL – the value itself. For repeated expressions braces could be used in the form $NUM_1 * \{NUM_2 * VAL\}$. For example, expression $2 * \{1\ 2 * \{3\ 4\}\}$ means that the following data set is defined: 1 3 4 3 4 1 3 4 3 4.

6.1   Definition section

Definition section contains initial data necessary for oil-and-gas reservoir model specification.

6.1.1 TITLE

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

Used to specify model title. Any alphabet can be used. The title should be one line long.

```
Example
TITLE
Test Example
/
```

This example specifies title for model "Test Example".

```
Example
TITLE
Model of field with large
number of blocks
/
```

This example specifies title for model "Model of field with large". Note that the information from the second line was ignored.

6.1.2 REPORTFILE / REPORTSCREEN

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keywords are used to specify reporting level for log file and for the screen. REPORTFILE and REPORTSCREEN keywords provide more advanced capabilities for tuning of output to log file, and to the screen. To provide most correct reporting on read input data, keywords REPORTFILE and REPORTSCREEN should be specified in the very beginning of `*.data`-file.

All output to the `log`-file and to the screen will be performed using unit system indicated in UNITS (see 6.1.10).

Note: METRIC unit system in tNavigator differs from some other simulators' unit systems in pressure units: tNavigator uses Atmospheres, not bars. For example, connection factor will be measured in $m^3/(day \cdot Atm)$, and not in $m^3/(day \cdot bar)$, like in Eclipse.

The following values should be specified:

- section, the following options are available:

\<Section\>	\<Description\>
READ	Information on reading input data
WRITE	Information on writing input data
CHECK	Information on checking input data
INIT	Information on initialization of simulation process
ITERS	Information on iteration process
FIP	Information on FIP
SOLVE	Information on solving
GUI	Information on user interface
RSV	Information on reservoir
GROUP	Information on well groups
WELL	Information on wells
CONN	Information on connections
GRID	Information on grid
PRINT_LOGS	All log messages and tables
ALL	All sections listed above

- priority, characterizing the amount of output to log file (to the screen). The following options are possible:

\<Priority\>	\<Description\>
WARNING	Report only warnings and errors
HIGHEST	Report information with highest priority and above
HIGH	Report information with high priority and above
MEDIUM	Report information with medium priority and above
LOW	Report all information

For example, for section PRINT_LOGS highest priority means information (i.e. tables) only about reservoir, high priority means information on reservoir and groups, medium priority also includes well information, and, finally, low priority means that all information on reservoir, groups, wells and connections will be written to log file (displayed on the screen). For other sections similar information priority system is used.

One or more pairs <section> <priority> should be specified after keyword REPORTFILE (or REPORTSCREEN). The list should be terminated by a slash /.

The effect of REPORTFILE and REPORTSCREEN keywords is illustrated in the following table. Having set a priority for a section, you will get all messages indicated for this section and this priority and messages to the right. If a section is not present in the table, this means that priorities choice will not reflect on output to `*.log` file and to the screen, since all messages for this section are warning and error messages.

Priority	LOW	MEDIUM	HIGH	HIGHEST	WARNING
READ	• Values of RUNCTRL parameters read	• 1 line for each keyword successfully read: its name and number of lines (values) read; for DIMENS, TABDIMS keywords the values are listed	• List of included *.inc-files and their names		• Error and warning messages
CHECK			• Messages on block deactivation: each block and reason for deactivation are listed	• Number of connections in the model	• Error and warning messages, for example, total number of blocks switched to inactive for each constraint type
INIT			• Total number of active blocks • Total memory allocated	• Number of blocks, switched to inactive due to minimal pore volume criteria, MINPV (see 6.2.23)	• Error and warning messages • Names of *.data, *.res, *.for, *.err, *.log files

to be continued on the next page

continued from previous page

Priority	LOW	MEDIUM	HIGH	HIGHEST	WARNING
ITERS		• Information on each simulator time step – time, step length, number of iterations, variable change – dp, ds (or dn), dv • In case of time step restart – corresponding information: restart moment, old and new time step lengths • Messages on Newton process convergence problems that cause restart – too slow convergence or convergence to nonphysical solution	• Restart time	• Message on simulation start date (SIMULATION STARTED FROM DATE...) • After each report time step – message on time step completion and summary – rates, average pressure (REPORT N .. FOR DAY .. IS DONE) • After simulation is finished – total elapsed time, number of time steps and iterations (Total elapsed ...)	• Error and warning messages
FIP		• Summary tables are enlarged with one column for each FIP region with the same data	• Summary tables for all FIP regions – reserves, average reservoir pressure, pore volume		• Error and warning messages

continued from previous page

Priority	LOW	MEDIUM	HIGH	HIGHEST	WARNING
SOLVE	• Information on each Newton iteration – number, residual, parameters change				• Error and warning messages
RSV	• Information on material balance error				• Error and warning messages
GROUP	• Information on group controls			• Message on compensation switch off, if total injection for group is too small	• Error and warning messages
WELL	• Messages on well switch • Messages on problems with well calculation			• Information on well / perforation closure due to economic limit on rate or watercut	• Error and warning messages

continued from previous page

Priority	LOW	MEDIUM	HIGH	HIGHEST	WARNING
CONN	• Messages on crossflow through connections				• Error and warning messages
GRID	• Messages on grid				• Error and warning messages
PRINT_LOGS	• Injection/production tables for each connection	• Injection/production tables for each well	• Injection/production tables for each group • All data on new well perforations	• Summary injection/production tables for reservoir	• Error and warning messages

Default:
REPORTFILE
ALL   HIGH
/
REPORTSCREEN
ALL   HIGH
PRINT_LOGS   WARNING
FIP   WARNING
/

Example
REPORTFILE
READ LOW
PRINT_LOGS MEDIUM
/
REPORTSCREEN
PRINT_LOGS MEDIUM
/

This example defines that all names and number of read variables will be written to log file, and all tables concerning production from reservoir, groups and wells will be written both to the screen and to the log file.

Example
REPORTFILE
ITERS HIGH
WELL LOW
/
REPORTSCREEN
WELL LOW
ITERS MEDIUM
/

This example defines that information on each calculation time step length, relative pressure and saturation variation after this time step and relative material balance error for this time step will be written to the screen. The information on well control change will also be written to the screen. In the log file there will be no information on calculation time steps, summaries only for reporting time steps will be accompanied by information on well control changes.

6.1.3 RPTRST

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword allows to save additional calculated data to results file (`*.rst`). List of mnemonics to be saved should be specified below. List should end with a slash (`/`).

The following mnemonics are accepted:

- FLOWW - accumulted interblock flows of water phase

- FLOWO - accumulted interblock flows of oil phase

- FLOWG - accumulted interblock flows of gas phase

Default: None.

```
Example
RPTRST
FLOWO FLOWG /
```

This example requests saving two additional parameter maps to results file – oil and gas accumulated flows.

6.1.4 OUTSOL

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar	
☑ eclipse 100	☐ CMG imex	☐ CMG stars	

The keyword is identical with the keyword RPTRST (see 6.1.3).

6.1.5   GRIDFILE

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines save of calculation results in EGRID and INIT format files (`*.EGRID` format and `*.INIT` format of Eclipse simulator). These files will be generated, if simulator (it's console version) is run with option `-b` ("b" stands for "binary").

The following parameters should be defined (list should end with a slash, `/`).

- this is an Eclipse compatibility field, possible values are 0, 1, 2, 1* (i.e. default); value of this parameter is IGNORED because of different processing of result files in tNavigator;

- parameter indicating whether files of INIT and EGRID format should be output (the latter containing data on inactive cells, non-neighbor connections, local grid refinements and coarsenings); possible values: 0 (don't output), 1 (output).

Default: output of EGRID and INIT files: 1 (i.e. output).

```
Example
GRIDFILE
2 1
/
```

In this example files of EGRID and INIT format will be written after calculation with `-b` option. File of `*.GRID` format will not be generated (first argument of keywrod **GRIDFILE** is ignored in current version of tNavigator).

6.1.6 SUMMARY

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines save of calculation results in `*.UNSMRY`, `*.SMSPEC` format files, containing production data (in format of Eclipse simulator). These files will be generated, if simulator (it's console version) is run with option `-b` ("b" stands for "binary").

Example
`SUMMARY`

In this example files of `*.UNSMRY`, `*.SMSPEC` format will be written after calculation with `-b` option.

6.1.7 START

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the start simulation date. The date should be written in format: DD 'MONTH' YYYY. The date should be terminated with a slash /. 'MONTH' can take the following values: JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC, or JLY, which is acceptable alternative to JUL.

Any report dates in the keyword DATES (see 6.10.33) must be later than the start date.

Example
START
01 JUL 1984
/

6.1.8 RESTART

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword is used for calculation restart basing on already calculated model from N-th time step. It is also possible to restart computation process from the date specified, see RESTARTDATE (see 6.1.9).

Restart data are read from the saved file `*.rst`. The following data should be specified:

- file name of `*.rst` file with saved data, file extension need not be specified,

- time step number from which computation should restart.

```
Example
RESTART
test1 10
/
WCONPROD
111 OPEN LRAT 29.29 0 0 29.29 2*
112 OPEN LRAT 32.63 0 0 32.63 2*
119 OPEN LRAT 53.2 0 0 53.2 2*
120 OPEN LRAT 46.27 0 0 46.27 2*
121 OPEN LRAT 12.24 0 0 12.24 2*
126 OPEN LRAT 24.63 0 0 24.63 2*
127 OPEN LRAT 46.27 0 0 46.27 2*
128 OPEN LRAT 18.26 0.7719 0 19.04 2*
129 OPEN LRAT 38.31 0 0 38.31 2*
202p OPEN LRAT 35.28 0 0 35.28 2*
/
TSTEP
30
/
```

In this example data are loaded from the file `test1.rst` and computation starts from 10-th time step (first step to be calculated will be step 11). Schedule section without leading keyword DATES should follow after RESTART.

6.1.9 RESTARTDATE

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used for calculation restart basing on already calculated model from the date specified. It is also possible to restart calculation from specified time step, see RESTART (see 6.1.8). Restart data are read from the saved file `*.rst`. The following data should be specified:

- file name of `*.rst` file with saved data; file extension should not be specified,

- `*` symbol, field is ignored;

- date from which computation should restart; if start of model time step does not coincide with this date, then computation starts from the nearest date before specified one;

```
Example
RESTARTDATE
test1 1* 10 NOV 1998
/
WCONPROD
111 OPEN LRAT 29.29 0 0 29.29 2*
112 OPEN LRAT 32.63 0 0 32.63 2*
119 OPEN LRAT 53.2 0 0 53.2 2*
120 OPEN LRAT 46.27 0 0 46.27 2*
121 OPEN LRAT 12.24 0 0 12.24 2*
126 OPEN LRAT 24.63 0 0 24.63 2*
127 OPEN LRAT 46.27 0 0 46.27 2*
128 OPEN LRAT 18.26 0.7719 0 19.04 2*
129 OPEN LRAT 38.31 0 0 38.31 2*
202p OPEN LRAT 35.28 0 0 35.28 2*
/
TSTEP
30
/
```

In this example data are loaded from the file `test1.rst` and computation starts from November 10, 1998. Schedule section without leading keyword DATES should follow after RESTART.

6.1.10 UNITS

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Units used in all input/output data (see 4) follow on the next string.
The following units systems are supported:

- METRIC – metric system

- FIELD – field system

Default: METRIC.

Example
UNITS
METRIC

In this example metric system is used.
Henceforward the examples of this manual use metric system.

6.1.11　LANGUAGE

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used for language and encoding used for input and output data. The following combination of language and encodings are supported:

- ENGLISH - English,

- RUS.WIN - Russian in cp1251 (windows) encoding,

- RUS.ALT - Russian in cp866 (DOS) encoding,

- RUS.KOI - Russian in KOI8 (UNIX) encoding.

The language and encoding specified are used:

- for input data,

- for generated by simulator files with reports,

- for all messages output on terminal.

Default: ENGLISH.

```
Example
LANGUAGE
RUS.ALT
```

In this example simulator will use Russian in DOS encoding.

6.1.12 DIMENS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines numbers of grid blocks NX, NY, NZ in X, Y, Z directions. X and Y are surface coordinates, and Z axis is directed vertically downward. The total number of cells is a product of these three numbers: NX*NY*NZ.

Default: 100 100 1

```
Example
DIMENS
50 50 1
/
```

This example defines grid having $50 * 50 * 1$ blocks, 50 blocks in X direction, 50 blocks in Y direction and 1 block in Z direction. The grid will consist of 2500 blocks.

```
Example
DIMENS
30 23 5
/
```

This example defines a $30 * 23 * 5$ grid, with 30 blocks in X direction, 23 blocks in Y direction and 5 blocks in Z direction. Total number of blocks is 3450.

6.1.13   TABDIMS

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The data consists of five items, describing number of regions with different parameter values. The numbers specify:

1. number of saturation function regions (SATNUM (see 6.4.2)),

2. number of regions with different PVT properties (PVTNUM (see 6.4.1)),

3. maximum number of nodes of saturation in one PVT table, entered by keywords SGFN (see 6.6.5), SGOF (see 6.6.2) (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

4. maximum number of nodes of pressure in one PVT table, entered by keywords PVTG (see 6.5.7), PVTO (see 6.5.3), PVDG (see 6.5.6), PVDO (see 6.5.1) (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

5. number of fluid-in-place regions (FIPNUM (see 6.4.4)).

6. maximum number of nodes of oil vaporization in gas $R_{G,O}$ in one VPT table, entered by keywords PVTO (see 6.5.3), PVCO (see 6.5.5). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

7. maximum number of nodes of oil-gas ratio $R_{O,G}$ in one PVT table of wet gas, entered by a keyword PVTG (see 6.5.7). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

8. maximum number of saturation end-point versus temperature tables (tables are specified using the keyword ENPTVT (see 6.8.65)).

9. maximum number of equation of state regions for reservoir conditions (these regions are entered using the keyword EOSNUM (see 6.4.10)).

10. maximum number of equation of state regions for surface conditions. Default: is equal to the 9-th parameter of this keyword.

11. maximum number of flux regions (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

12. maximum number of thermal regions (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

13. number of rock tables (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

14. maximum number of pressure maintenance regions (IGNORED). This is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

15. the number of temperatures for $K_i(p, T)$ tables (KVTEMP (see 6.8.5), KVTABTn (see 6.8.6)).

The division of grid blocks into regions and setting of appropriate values should be specified below.

Default: 1 1 20 20 1 20 20 1 1

Example
TABDIMS
2 4 2* 3
/

In this example the grid representing the reservoir is

- divided into two regions with different flow characteristics,

- divided into four PVT regions, each of them with different PVT properties tables,

- 3-rd and 4-th numbers are equal 20 (default)

- divided into three FIP regions, for each of them fluid-in-place data will be reported.

6.1.14 EQLDIMS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies dimensions of the equilibration tables. The data should be terminated with a slash /.

1. number of equilibration regions (EQLNUM (see 6.4.3), EQUIL (see 6.9.1));

2. number of depth nodes in any table of pressure versus depth constructed internally by the equilibratio algorithm; IGNORED, this is an Eclipse compatibility field; In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

3. maximum number of depth nodes in any table RSVD (see 6.9.2), RVVD (see 6.9.4); IGNORED, this is an Eclipse compatibility field; In tNavigator number of lines does not have to be specified because the tables are allocated dynamically,

4. maximum number of tables of initial tracer concentration versus depth (TVDP (see 6.9.26));

5. maximum number of lines in the table of initial tracer concentration versus depth (TNUM (see 6.9.25)).

Default:

1. 1;

2. 100;

3. 20;

4. 1;

5. 20.

Example
```
EQLDIMS
3 2* 2 5
/
```

In this example the grid will be divided into three regions, for each of them initial distribution of saturations and pressure will be calculated basing on thermodynamic equilibrium. Maximum number of tables of initial tracer concentration versus depth – 2 tables. Maximum number of lines in the table of initial tracer concentration versus depth – 5 lines.

6.1.15 ACTDIMS

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets dimensions for the keywords ACTION (see 6.10.51), ACTIONW (see 6.10.54), ACTIONG (see 6.10.52), ACTIONR (see 6.10.53). The data should be terminated with a slash /.

1. the maximum number of types of keywords ACTION (see 6.10.51); ignored, this is an Eclipse compatibility field. In tNavigator number of types does not have to be specified because the memory is allocated dynamically,

2. the maximum number of lines of SHEDULE data (including the ENDACTIO keyword), that can be defined in any one ACTION (see 6.10.51); ignored, this is an Eclipse compatibility field. In tNavigator number of lines does not have to be specified because the memory is allocated dynamically,

3. the maximum number of characters per line of ACTION (see 6.10.51) data; ignored, this is an Eclipse compatibility field. In tNavigator number of characters does not have to be specified because the memory is allocated dynamically,

Default:

1. 2;

2. 50;

3. 80.

Example
```
ACTDIMS
5 40 100
/
```

6.1.16   REGDIMS

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The data consists of four items, describing number of regions with different parameter values. The numbers specify:

- 1-st number — number of fluid-in-place regions (FIPNUM (see 6.4.4)), also may be specified with keyword TABDIMS (see 6.1.13),

- 2-nd number — number of sets of fluid-in-place regions (FIPNUM (see 6.4.4)),

- 3-rd number — number of independent reservoir regions,

- 4-th number — number of flux regions (FLUXNUM (see 6.4.8)),

The division of grid blocks into regions and setting of appropriate values should be specified below.

Default: 1 1 0 0

```
Example
REGDIMS
2 2 0 0
/
```

In this example the grid representing the reservoir is

- divided into two FIP regions, for each of them fluid-in-place data will be reported,

- divided into two sets of FIP regions, for each of them fluid-in-place data will be reported.

6.1.17 VFPIDIMS

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the dimensions of BHP tables for injectors, which are specified using the keyword VFPINJ (see 6.10.18). The data should be terminated with a slash /.

The following parameters are to be specified:

1. maximum number of flow values (FLO) in each table;

2. maximum number of tubing head pressure values (THP) in each table;

3. maximum number of BHP tables for injectors.

Default: 0 0 0

```
Example
VFPIDIMS
10 5 3
/
```

This example sets: maximum number of flow values (FLO) in each table – 10, maximum number of tubing head pressure values (THP) in each table – 5, maximum number of BHP tables for injectors – 3.

6.1.18 VFPPDIMS

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets the dimensions of BHP tables for producers, which are specified using the keyword VFPPROD (see 6.10.19). The data should be terminated with a slash /.

The following parameters are to be specified:

1. maximum number of flow values (FLO) in each table;

2. maximum number of tubing head pressure values (THP) in each table;

3. maximum number of water fraction (WFR) in each table;

4. maximum number of gas fraction (GFR) in each table;

5. IGNORED, this is an Eclipse compatibility field;

6. maximum number of BHP tables for producers.

Default: 0 0 0 0 0

```
Example
VFPPDIMS
8 5 3 1 0 2
/
```

This example sets: maximum number of flow values (FLO) in each table – 8, maximum number of tubing head pressure values (THP) in each table – 5, maximum number of water fraction (WFR) in each table – 3, maximum number of gas fraction (GFR) in each table – 1, maximum number of BHP tables for producers – 2.

6.1.19   ROCKCOMP

√	tNavigator	√	eclipse 300	☐	MORE Roxar
√	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword sets the rock compaction option and maximum number of rock regions (tables of transmissibility dependence on pressure).

The are three parameters (The data should be terminated with a slash /.):

1. rock compaction hysteresis or irreversibility option:

   - REVERS – rock compaction is fully reversible with increasing pressure;
   - IRREVERS – rock compaction is irreversible with increasing pressure (the pore space does not re-inflate).

2. the maximum number of rock regions,

3. flag of the Water Induced Compaction option is used: YES or NO. (IGNORED), this is an Eclipse compatibility field.

Tables of transmissibility dependence on pressure for each rock region are set using the keyword ROCKTAB (see 6.2.10). For each grid block one should specify the rock region to which it belongs using ROCKNUM (see 6.4.6).

Example
```
ROCKCOMP
IRREVERS 5
/
```

In this example there are 5 rock regions and rock compaction is irreversible.

6.1.20   ROCKDIMS

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword specifies the number of types of cap and base rocks with different properties, which will be used to model the heat exchange between the reservoir and surroundings. The data should be terminated with a slash /.

The are three parameters:

1. maximum number of types of cap and base rocks with different properties,

2. IGNORED,

3. the maximum number of grid blocks which can be connected to any rock type (this parameter is used in the keyword ROCKCON (see 6.2.58)).

   Default:
The value of 3-rd parameter is equal to the value of 1-st parameter.

```
Example
ROCKDIMS
2 1* 6
/
```

In this example the maximum number of types of cap and base rocks with different properties is 2; the maximum number of grid blocks which can be connected to any rock type is 6.

6.1.21 WELLDIMS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets up dimensions of the well data. The data should be terminated by a slash `/`.

12 parameters are to be specified:

1. the maximum number of wells,

2. the maximum number of connections that one well can have,

3. the maximum number of well groups in the model,

4. the maximum number of wells in one group,

5. the maximum number of separator stages (the last one - the stock tank - is included),

6. the maximum number of well streams,

7. the maximum number of mixtures (using in injectors),

8. the maximum number of separators (field separator is included),

9. the maximum number of items in any mixture (using in injectors),

10. the maximum number of connections groups allowed to cross domain boundary of one process during a parallel run,

11. the maximum number of the well lists that one well can belong to simultaneously,

12. the maximum number of dynamic well lists during the run.

Default: any parameter — zero.

```
Example
WELLDIMS
200 7 10 20 5 10 5 4 2 0 1 1
/
```

6.1.22   WSEGDIMS

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets dimensions for multi-segment wells (section 2.15.2). The data should be terminated with a slash /.

The following data should be entered:

1. maximum number of multi-segment wells; (IGNORED). This is an Eclipse compatibility field. In tNavigator number of does not have to be specified because the memory is allocated dynamically;

2. maximum number of segments per well; (IGNORED). This is an Eclipse compatibility field. In tNavigator number of does not have to be specified because the memory is allocated dynamically;

3. maximum number of branches per multi-segment well; (IGNORED). This is an Eclipse compatibility field. In tNavigator number of does not have to be specified because the memory is allocated dynamically.

Example
WSEGDIMS
10 20 4/

6.1.23   TRACERS

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets dimensions for tracers. The data should be terminated with a slash /.

1. the maximum number of passive oil tracers (entered via the keyword TRACER (see 6.5.24)); ignored, this is an Eclipse compatibility field,

2. the maximum number of passive water tracers (entered via the keyword TRACER (see 6.5.24)); ignored, this is an Eclipse compatibility field,

3. the maximum number of passive gas tracers (entered via the keyword TRACER (see 6.5.24)); ignored, this is an Eclipse compatibility field,

4. the maximum number of environmental tracers (for environmental tracer option); ignored, this is an Eclipse compatibility field,

5. ignored, this is an Eclipse compatibility field,

6. ignored, this is an Eclipse compatibility field,

7. ignored, this is an Eclipse compatibility field,

8. ignored, this is an Eclipse compatibility field,

9. ignored, this is an Eclipse compatibility field,

10. ignored, this is an Eclipse compatibility field,

11. ignored, this is an Eclipse compatibility field,

12. ignored, this is an Eclipse compatibility field.

```
Example
TRACERS
4 5
/
```

In this example there are: 4 passive oil tracers and 5 passive water tracers.

6.1.24 NONNC

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

If this keyword is used, non-neighbor connections won't be allowed. By default non-neighbor connections are allowed.

Such connections may be set differently, including explicit definition using keyword NNC (see 6.2.37).

Example
```
NONNC
```

In this example non-neighbourhood connections are not allowed.

6.1.25 THERMAL

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to label that the thermal simulatin will be used. The keywords which can be used in thermal simulation are discribed in the section "Thermal properties" (6.8).

Example
```
THERMAL
```

6.1.26 REACTION

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies the number of chemical reactions (n_r). The data should be terminated with a slash /.

```
Example
REACTION
3 /
```

In this example there are 3 chemical reactions.

6.1.27 OIL

	tNavigator		eclipse 300		MORE Roxar
√		□		□	
√	eclipse 100	□	CMG imex	□	CMG stars

The keyword is used to label that a model contains an oil phase. The keyword should be used if an oil phase exist or could exist in the model. OIL is added automatically for any compositional model. No parameters are to be specified.

6.1.28 GAS

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to label that a model contains a gas phase. The keyword should be used if a gas phase exist or could exist in the model. GAS is added automatically for any compositional model. No parameters are to be specified.

6.1.29 WATER

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to label that a model contains a water phase (active). No parameters are to be specified.

6.1.30 VAPOIL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword is used to label that there is vaporised oil in a wet gas. One can use this keyword only if OIL (see 6.1.27) and GAS (see 6.1.28) are specified.

Let's consider two conditions: vaporised oil has constant and uniform concentration in a gas phase and the pressure is always above a dew point. In that case a run can be more efficiency in the following way:
OIL (see 6.1.27) and VAPOIL shouldn't be specified; gas should be considarated as a dry gas; specify vaporised oil concentration using keyword RVCONST (see 6.5.11) in the GRID (6.2) section.

The model will be considered as a model with dry gas, but the constant vaporised oil concentration will be taken into consideration during a simulation.

No parameters are to be specified.

6.1.31 DISGAS

√ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
√ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword is used to label that there is a dissolved gas in live oil. One can use this keyword only if OIL (see 6.1.27) and GAS (see 6.1.28) are specified.

Let's consider two conditions: dissolved has constant and uniform concentration in an oil phase and the pressure is always above a dew point. In that case a run can be more efficiency in the following way:

GAS (see 6.1.28) and DISGAS shouldn't be specified; oil should be considarated as a dead oil; specify dissolved gas concentration using keywords RSCONST (see 6.5.9), RSCONSTT (see 6.5.10) in the GRID (6.2) section.

The model will be considered as a model with dead oil, but the constant dissolved gas concentration will be taken into consideration during a simulation.

No parameters are to be specified.

6.1.32 SOLID

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword is used to label that there is a solid phase in the model (solid phase consists of one component – coke).

6.1.33   BRINE

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword indicates that the Brine Traking option is enable, to allow the modeling of waters with different salinities.

6.1.34 NUMRES

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets a number of reservoirs (if the is more than one reservoir). NUMRES should be present if there is more than one set of coordinate lines (are used in keywords COORD (see 6.2.32), ZCORN (see 6.2.33)). If the is more than one reservoir the same number of sets of data COORD should be specified using the keyword RESVNUM (see 6.2.34) and the bounds of reservoirs should be specified using the keyword COORDSYS (see 6.2.35). The data should be terminated with a slash /.

Default: 1

Example
```
NUMRES
3
/
```

6.1.35   INCLUDE

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used in the keyword-based file to include another keyword-based file. The included file may, in turn, contain INCLUDE keywords. The nesting level is not limited.

If included file name contains spaces, it should be surrounded by quotation marks. Only one file may be specified after INCLUDE keyword.

```
Example
INCLUDE title_grid.inc
/
```

This example causes the program to include file `title_grid.inc` in the analyzed file. Note that file name does not contain spaces.

```
Example
INCLUDE "title grid.inc"
/
```

```
Example
INCLUDE 'title grid.inc'
/
```

These two example cause the program to include the same file `title grid.inc`. Note that file name contains a space.

6.1.36 IMPLICIT

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword is used to set the implicit solution procedure for the simulation.

Example
```
IMPLICIT
```

6.1.37 DUALPORO

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets that the Dual porosity option (2.20) will be used in the run.

Dual porosity. In a resevoir with the dual porosity there are two systems: rock matrix (the biggest part of the reservoir) and fractures (which have high permeability).

Dual porosity single permeability: fluid flow between matrix cells is possible only using fractures. Fluid flow through the reservoir is possible only in fractures.
Dual porosity dual permeability: fluid flow between neighboring matrix cells is possible.

Dual porosity is specifying by the keyword DUALPORO.
Dual permeability — DUALPERM (see 6.1.38).

If these options are used, for every geometric grid block we consider two cells: the matrix part and the fracture part of this block. One can specify their properties (porosity, permeability etc.) independently.

If the keyword DUALPORO or DUALPERM (see 6.1.38) is used, the number of layers in the Z-direction should be even (this number is entered by the third parameter of the keyword DIMENS (see 6.1.12) (NZ).
The first half of the grid blocks corresponds to the matrix cells, and the second half – fracture cells. tNavigator autimatically create non-neighbor connections which correspond to the matrix-fracture flows.

Example
```
DUALPORO
```

6.1.38 DUALPERM

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**	
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**	

    The keyword sets that the Dual permeability option will be used in the run. This option allows fluid flows between matrix cells (2.20).

Example
```
DUALPERM
```

6.1.39 LGR

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets options and dimensions for local grid refinement (LGR) – section 2.21. The data should be terminated with a slash /.

The following parameters should be specified:

1. maximum number of LGRs in the model. This is an Eclipse compatibility field. (IGNORED) In tNavigator specification of this number is not obligatory, because memory is allocated dynamically;

2. maximum number of cells in each LGR. This is an Eclipse compatibility field. (IGNORED) In tNavigator specification of this number is not obligatory, because memory is allocated dynamically;

3. maximum number of amalgamated coarse cells. This is an Eclipse compatibility field. (IGNORED) In tNavigator specification of this number is not obligatory, because memory is allocated dynamically;

4. maximum number of LGR amalgamations. This is an Eclipse compatibility field. (IGNORED) In tNavigator specification of this number is not obligatory, because memory is allocated dynamically;

5. maximum number of LGRs in any amalgamation. This is an Eclipse compatibility field. (IGNORED) In tNavigator specification of this number is not obligatory, because memory is allocated dynamically;

6. This is an Eclipse compatibility field. (IGNORED)

7. This is an Eclipse compatibility field. (IGNORED)

8. This is an Eclipse compatibility field. (IGNORED)

6.1.40 VISCD

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets that the Viscous displacement option will be used in the dual porosity run (2.20).

If this option is used, one should specify the distances between fractures (matrix block sizes) in X, Y and Z directions using keywords LX (see 6.2.50), LY (see 6.2.50), LZ (see 6.2.50).

Viscous Displacement. Viscous displacement – fluid flow under the influence of pressure gradient. One can observe a pressure gradient in the dual porosity system. This gradient moves the fluid in the fracture towards the production well. If this gradient is small and fracture permeability is high, the matrix-fracture viscous displacement under the influence of pressure gradient isn't considered. Nevertheless, if fractions have small permeability then the matrix-fracture viscous displacement under the influence of pressure gradient can be very important in production.

tNavigator can compute a SIGMA (see 6.2.53) factor using keywords LX (see 6.2.50), LY (see 6.2.50), LZ (see 6.2.50) and LTOSIGMA (see 6.2.55). If tNavigator compute SIGMA, any manually input of this parameter (SIGMA (see 6.2.53)) in data file will be ignored.

Example
```
VISCD
```

6.1.41 NODPPM

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword NODPPM cancels a multiplication of permeability (for the fracture blocks) by porosity (fracture blocks) during the dual porosity run (2.20). Since this multiplication is used to obtain a net bulk fracture permeability one have to enter this value manually if NODPPM is enable.

Example
```
NODPPM
```

6.1.42 NETWORK

√ tNavigator	√ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets dimensions for extended network model. The extended network model is specified via keywords NODEPROP (see 6.10.27), BRANPROP (see 6.10.26). The data should be terminated with a slash /.

(Section: NETWORK option. Automatic chokes. Compressors. – 2.16.)

The following parameters should be specified:

- maximum number of nodes in the network model;

- maximum number of branches in the network model;

- maximum number of branches connected to any one node in the network model.

Example
```
NETWORK
10 9 2 /
```

In this example:

- maximum number of nodes in the network model – 10;

- maximum number of branches in the network model – 9;

- maximum number of branches connected to any one node in the network model – 2.

6.1.43 END

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword terminates the input of data in data file. This keyword doesn't have any parameters.

Example
END

6.2   Grid section

This section describes all keywords necessary for rectangular grid hydrodynamic model description.

In tNavigator local grid refinements can be specified – section 2.21.

6.2.1 DX / DY / DZ

✓	**tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keywords specify sizes of cells first in *X* direction, then in *Y* and *Z* directions. The same number of values as number of cells in corresponding direction must be specified (NX * NY * NZ).

Default: none

```
Example
DIMENS
10 10 2
/
...
DX
50*610 150*520
/
DY
200*470
/
DZ
100*5
100*10
/
```

This example sets grid block sizes for a 200-block grid: first 50 grid blocks are $610*470*5$ m^3, next 50 grid blocks are $520*470*5$ m^3, and the rest 100 grid blocks are $520*470*10$ m^3.

In the three consecutive examples NX=5, NY=3, NZ=2.

```
Example
DIMENS
5 3 2
/
...
DX
30*200
/
```

In this example all grid block sizes in *X* direction are the same and are equal to 200 m.

Example
DIMENS
5 3 2
/
...
DX
6*{1*100 4*200}
/

In this example for a 5x3x2 grid X-direction grid block sizes are set, here cells with coordinates (1,1,1), (1,2,1), (1,3,1), (1,1,2), (1,2,2), (1,3,2) have DX=100, and the rest of them have DX=200.

Example
DIMENS
5 3 2
/
...
DZ
6.84 6.62 6.87 6.95 6.99 6.80 6.94 7.70 7.76
8.40 7.70 7.76 6.93 8.83 10.63
6.84 6.62 6.87 6.95 6.99 6.80 6.94 7.70 7.76
8.40 7.70 7.76 6.93 8.83 10.63
/

In this example sizes of all grid blocks in Z direction are specified.

6.2.2 DXV

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

Used to set grid block sizes in X direction, in case when all blocks with the same i-coordinate have the same size in this direction. NX values should be specified, where NX is set in DIMENS (see 6.1.12).

Default: None.

```
Example
DIMENS
12 10 2
/
DXV
10 3*9 2*2 10 3*9 2*2 /
```

In this example all cells with i-coordinate equal to 1 have $DX = 10$; cells with i-coordinates 2, 3, 4 have $DX = 9$, cells with i-coordinates 5, 6 have $DX = 2$. Moreover, cells with i-coordinate 7 have $DX = 10$, with coordinates 8, 9, 10 have $DX = 9$, and with coordinates 11, 12 have $DX = 2$.

The same distribution of grid cells DX dimensions may be set through

```
Example
DIMENS
12 10 2
/
DXV
2*{10 3*9 2*2} /
```

6.2.3   DYV

√ **tNavigator**	√ **eclipse 300**	☐ **MORE Roxar**
√ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

Used to set grid block sizes in Y direction, in case when all blocks with the same j-coordinate have the same size in this direction. NY values should be specified, where NY is set in DIMENS (see 6.1.12).

Default: None.

```
Example
DIMENS
12 12 2
/
DYV
10 3*9 2*2 10 3*9 2*2 /
```

In this example all cells with j-coordinate equal to 1 have $DY = 10$; cells with j-coordinates 2, 3, 4 have $DY = 9$, cells with j-coordinates 5, 6 have $DY = 2$. Moreover, cells with j-coordinate 7 have $DY = 10$, with coordinates 8, 9, 10 have $DY = 9$, and with coordinates 11, 12 have $DY = 2$.

The same distribution of grid cells DY dimensions may be set through

```
Example
DIMENS
12 12 2
/
DYV
2*{10 3*9 2*2} /
```

6.2.4   DZV

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

Used to set grid block sizes in Z direction, in case when all blocks with the same k-coordinate have the same size in this direction. NZ values should be specified, where NZ is set in DIMENS (see 6.1.12).

Default: None.

```
Example
DIMENS
12 10 12
/
DZV
10 3*9 2*2 10 3*9 2*2 /
```

In this example all cells with k-coordinate equal to 1 have $DZ = 10$; cells with k-coordinates 2, 3, 4 have $DZ = 9$, cells with k-coordinates 5, 6 have $DZ = 2$. Moreover, cells with k-coordinate 7 have $DZ = 10$, with coordinates 8, 9, 10 have $DZ = 9$, and with coordinates 11, 12 have $DZ = 2$.

The same distribution of grid cells DZ dimensions may be set through

```
Example
DIMENS
12 10 12
/
DZV
2*{10 3*9 2*2} /
```

6.2.5   TOPS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Depths of the upper layer cells follow on the next string. NX * NY values must be specified.
Default: none

```
Example
DIMENS
7 4 3
/
...
TOPS
2301.98 2303.87 2305.78 2306.86 2303.06 2301.29 2298.46
2297.36 2297.32 2294.87 2293.37 2293.87 2293.40 2296.34
2303.12 2302.26 2300.51 2300.07 2294.68 2288.54 2286.85
2238.09 2238.09 2238.09 2238.09 2301.15 2302.94 2304.96
/
```

This example sets depths for a 7x4x3 grid.

```
Example
DIMENS
10 10 1
/
...
TOPS
2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0
2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0
2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2294.0 2295.3
2296.2 2297.5 2298.9 2299.0 2297.7 2296.3 2294.9 2294.1
2294.8 2296.7 2298.7 2300.5 2300.3 2298.3 2296.7 2295.1
2293.6 2293.1 2293.0 2292.3 2291.5 2290.6 2290.4 2290.2
2289.2 2288.2 2287.1 2285.3 2281.3 2275.7 2270.1 2265.4
2266.0 2272.7 2281.5 2238.0 2238.0 2238.0 2238.0 2238.0
2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0
2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0 2238.0
2292.3 2293.5 2294.7 2295.9 2297.1 2298.2 2298.2 2296.9
2295.5 2294.0 2292.7 2293.0 2295.0 2297.0 2298.9 2299.6
2298.6 2297.1 2295.6 2294.2
/
```

This example sets depths of all upper layer cells. This layer consists of 100 blocks.

6.2.6 DEPTH

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify the depth of the center of the grid block. The keyword should be followed by one non-negative real number for every grid block.

Default: none.

```
Example
DIMENS
3 4 2
/
...
DEPTH
2301.98 2303.87 2305.78 2306.86 2303.06 2301.29
2297.36 2297.32 2294.87 2293.37 2293.87 2293.40
2303.12 2302.26 2300.51 2300.07 2294.68 2288.54
2238.09 2238.09 2238.09 2238.09 2301.15 2302.94
/
```

This example sets depths of the center of the blocks for the grid 3x4x2.

6.2.7   PERMX / PERMY / PERMZ

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

These keywords specify absolute permeability values first in X, then in Y and then in Z directions. The same number of values as number of grid blocks must be specified. These are diagonal components of absolute permeability tensor in 2.4.

Default: none

```
Example
DIMENS
5 5 4
/
...
PERMX
100*1
PERMY
100*1
PERMZ
100*10
```

In this example absolute permeability values in X (PERMX), Y (PERMY), and Z (PERMZ) directions are specified for a 100-block grid.

```
Example
DIMENS
5 6 2
/
...
PERMY
2*{5*103 5*104 5*105 15*106}
/
```

This example specifies Y-direction absolute permeability distribution using brackets and asterisk for more compact input. The defined permeability distribution is equivalent to the one in the next example.

Example
```
DIMENS
5 6 2
/
...
PERMY
103 103 103 103 103 104 104 104 104 104 105 105 105 105 105
106 106 106 106 106 106 106 106 106 106 106 106 106 106 106
103 103 103 103 103 104 104 104 104 104 105 105 105 105 105
106 106 106 106 106 106 106 106 106 106 106 106 106 106 106
/
```

6.2.8   PERMXY / PERMYZ / PERMXZ

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

These are off-diagonal components of absolute permeability tensor in 2.4. The permeability tensor is defined in the cartesian x, y, z axes. Tensor is assumed to be symmetric. $K_{11} = PERMX, K_{22} = PERMY, K_{33} = PERMZ, K_{12} = K_{21} = PERMXY, K_{23} = K_{32} = PERMYZ, K_{13} = K_{31} = PERMXZ$.

The same number of values as number of grid blocks must be specified. The data should be terminated with a slash /.

Not entered keywords are set to zero.

```
Example
DIMENS
5 5 4
/
...
PERMXY
20*100 40*150 40*120
/
```

In this example PERMXY is equal to 100 in first 20 blocks, in next 40 blocks - 150, and in last 40 blocks - 120.

6.2.9   RKTRMDIR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the number of parameters in one table row of the keyword ROCKTAB (see 6.2.10). If this keyword is present, there are 5 parameters, else — 3 parameters. In case of 3 parameters the third parameter is the transmissibility multiplier. In case of 5 parameters: the third is the multiplier in the X-direction, the fouth - in the Y-direction and the fifth - in the Z-direction.

Example
RKTRMDIR

6.2.10   ROCKTAB

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to set tables of transmissibility dependence on pressure for each rock region. Each table should be terminated with a slash /. The number of tables should be less of equal to the second parameter of the keyword ROCKCOMP (see 6.1.19).

The following parameters are to be specified (one table row):

1. pressure, values should increase down the column except for **STRESS** option (the 1-st parameter of the keyword ROCKOPTS (see 6.5.13)). In this case pressure values should decrease. If the overburden pressure ($P_{overburden}$ – OVERBURD (see 6.5.14)) is greater than the fluid pressure (P_{fluid}) (which is often true), the effective fluid pressure will be negative ($P_{effluid} = P_{fluid} - P_{overburden}$, $P_{effluid}$ - effective fluid pressure). In this case the rock compaction tables (ROCKTAB (see 6.2.10)) will have to be entered with negative pressure values. One can avoid this using **STRESS** option. Pressure values in this case can be entered positive: pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, which will be positive ($P_{efover} = P_{overburden} - P_{fluid}$).

2. porosity multiplier;

3. transmissibility multiplier (if RKTRMDIR (see 6.2.9) is present, this parameter is the transmissibility multiplier in the X-direction);

4. transmissibility multiplier in the Y-direction (can be specified only if RKTRMDIR (see 6.2.9) is present);

5. transmissibility multiplier in the Z-direction (can be specified only if RKTRMDIR (see 6.2.9) is present);

Default: values of transmissibility multiplier are linearly interpolated using pressure values; all values of multiplier but the ones on the first and last line may be defaulted.

```
Example
ROCKTAB
  21.74  0.9818  0.8915
  65.22  0.9850  1*
 108.70  0.9883  0.9341
 195.65  0.9948  0.9768
 282.61  1.0013  1.0194
 /
```

This example sets one table of transmissibility dependence on pressure for one rock region. Here transmissibility multipliers for a range of pressures are specified; value on the second line will be interpolated using pressure values and multiplier values from the first and third line.

6.2.11 MULTX

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to specify transmissibility multipliers in X-direction for plane between blocks $[I, J, K]$ and $[I+1, J, K]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTX
100*1.5
/
MULTX
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in X-direction as $1.5 * 2.0 = 3.0$.

6.2.12 MULTX-

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to specify transmissibility multipliers in X-direction for plane between blocks $[I, J, K]$ and $[I-1, J, K]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

If MULTX and MULTX- are both specified their product will be used. For example, MULTX is specified for block $[I, J, K]$ and MULTX- is specified for block $[I+1, J, K]$. Then the transmissibility multiplier between these two blocks is equal to the product of MULTX and MULTX-.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTX-
100*1.5
/
MULTX-
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in X-direction as $1.5 * 2.0 = 3.0$.

6.2.13   MULTY

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to specify transmissibility multipliers in Y-direction for plane between blocks $[I, J, K]$ and $[I, J+1, K]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTY
100*1.5
/
MULTY
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in Y-direction as $1.5 * 2.0 = 3.0$.

6.2.14 MULTY-

☑	tNavigator	☑	eclipse 300	☐	MORE Roxar
☑	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword is used to specify transmissibility multipliers in Y-direction for plane between blocks $[I,J,K]$ and $[I,J-1,K]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

If MULTY and MULTY- are both specified their product will be used. For example, MULTY is specified for block $[I,J,K]$ and MULTY- is specified for block $[I,J+1,K]$. Then the transmissibility multiplier between these two blocks is equal to the product of MULTY and MULTY-.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTY-
100*1.5
/
MULTY-
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in Y-direction as $1.5 * 2.0 = 3.0$.

6.2.15 MULTZ

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to specify transmissibility multipliers in Z-direction for plane between blocks $[I, J, K]$ and $[I, J, K+1]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTZ
100*1.5
/
MULTZ
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in Z-direction as $1.5 * 2.0 = 3.0$.

6.2.16 MULTZ-

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to specify transmissibility multipliers in Z-direction for plane between blocks $[I, J, K]$ and $[I, J, K-1]$. The keyword should be followed by one non-negative real number for every grid block. If this keyword is specified more than once, the old values are multiplied by the new ones.

If MULTZ and MULTZ- are both specified their product will be used. For example, MULTZ is specified for block $[I, J, K]$ and MULTZ- is specified for block $[I, J, K+1]$. Then the transmissibility multiplier between these two blocks is equal to the product of MULTZ and MULTZ-.

Default: 1 in all cells.

```
Example
DIMENS
5 5 4
/
...
MULTZ-
100*1.5
/
MULTZ-
100*2.0
/
```

In this example transmissibility multipliers are specified for all planes in Z-direction as $1.5 * 2.0 = 3.0$.

6.2.17 PORO

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword is used to define porosity for all grid blocks. The same number of values as number of grid cells must be specified. This is function $\phi(x,y,z)$ in 2.5.

Default: none

Example
```
DIMENS
5 5 4
/
...
PORO
100*0.1784
/
```

In this example for all 100 grid blocks the same porosity = 0.1784 is specified.

Example
```
DIMENS
5 6 2
/
...
PORO
0.20 0.20 0.20 0.20 0.20 0.19 0.19 0.19 0.19 0.19
0.18 0.18 0.18 0.18 0.18 0.17 0.17 0.17 0.17 0.17
0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17
0.20 0.20 0.20 0.20 0.20 0.19 0.19 0.19 0.19 0.19
0.18 0.18 0.18 0.18 0.18 0.17 0.17 0.17 0.17 0.17
0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17 0.17
/
```

This example defines porosity values for a 5x6x2 grid. By using the braces and asterisk, it is possible to define the same porosity distribution by means of the following expression.

Example
```
DIMENS
5 6 2
/
...
PORO
2*{5*0.2 5*0.19 5*0.18 15*0.17}
/
```

6.2.18 NTG

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword defines net to gross values of grid blocks. The same number of values as number of cells must be specified. This is function $\psi(x,y,z)$ in 2.5.

Default: 1 in all grid blocks

```
Example
DIMENS
5 5 4
/
...
NTG
100*0.2784
/
```

This example defines equal net to gross values = 0.2784 for all 100 blocks of the grid.

```
Example
DIMENS
5 4 2
/
...
NTG
0.60 0.60 0.60 0.60 0.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
0.60 0.60 0.60 0.60 0.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
/
```

This example defines net to gross values distribution for a 5x4x2 grid. By using the braces and asterisk, it is possible to define the same net to gross distribution by means of the following expression.

```
Example
DIMENS
5 4 2
/
...
NTG
2*{5*0.6 15*0.62}
/
```

6.2.19 DZNET

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines net thickness of grid blocks. The same number of values as number of cells must be specified. If DZNET is specified, net to gross ratio (function $\psi(x,y,z)$ in 2.5) will be calculated as follows:

$$NTG = \frac{DZNET}{DZ}$$

Default: *DZ* in all grid blocks

```
Example
DIMENS
5 5 4
/
...
DZ
100*5
/
DZNET
100*2.5
/
```

This example defines equal net thickness values = 2.5 for all 100 blocks of the grid. Net to gross ratio will be calculated as $\frac{2.5}{5} = 0.5$ for all blocks.

```
Example
DIMENS
5 4 2
/
...
DZNET
3.60 3.60 3.60 3.60 3.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
3.60 3.60 3.60 3.60 3.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
/
```

This example defines net thickness distribution for a 5x4x2 grid. By using the braces and asterisk, it is possible to define the same net thickness distribution by means of the following expression.

Example
```
DIMENS
5 4 2
/
...
DZNET
2*{5*3.6 15*0.62}
/
```

6.2.20 PORV

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines pore volumes of grid blocks. The same number of values as number of cells must be specified. If PORV is specified, net block pore volumes will be recalcuated.

Default: Calculated from grid block dimensions, PORO (see 6.2.17) and NTG (see 6.2.18).

```
Example
DIMENS
5 5 4
/
...
PORV
100*5000
/
```

This example defines block pore volume values = 5000 for all 100 blocks of the grid.

```
Example
DIMENS
5 4 2
/
...
PORV
3.60 3.60 3.60 3.60 3.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
3.60 3.60 3.60 3.60 3.60 0.62 0.62 0.62 0.62 0.62
0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62 0.62
/
```

This example defines pore volumes distribution for a 5x4x2 grid. By using the braces and asterisk, it is possible to define the same net thickness distribution by means of the following expression.

```
Example
DIMENS
5 4 2
/
...
PORV
2*{5*3.6 15*0.62}
/
```

Below pore volumes are defined for only part of the grid, using keyword ARITHMETIC (see 6.3.1).

```
Example
ARITHMETIC
PORV(1:5, 2:3, 7:7) = PORV/10
PORV(1:5, 2:3, 8:8) = 0
/
```

Here pore volumes of 10 grid blocks from 7th depth layer are divided by ten, and 10 blocks from next layer are deactivated by setting zero pore volume.

6.2.21 MULTPV

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines multiplers for pore volumes of grid blocks, calculated by simulator from grid block dimensions, PORO (see 6.2.17) and NTG (see 6.2.18).

The same number of values as number of cells must be specified. MULTPV keyword will have no effect on grid block pore volume if PORV (see 6.2.20) is specified for the same grid block. Otherwise net block pore volume will be recalcuated on the basis of MULTPV.

Default: 1.

```
Example
DIMENS
5 5 3
/
...
MULTPV
200 200 200 200 200
200 1 1 1 200
200 1 1 1 200
200 1 1 1 200
200 200 200 200 200
200 200 200 200 200
200 1 1 1 200
200 1 1 1 200
200 1 1 1 200
200 200 200 200 200
200 200 200 200 200
200 1 1 1 200
200 1 1 1 200
200 1 1 1 200
200 200 200 200 200

/
```

This example defines block pore volume multiplers equal to 200 on reservoir sides and unit in the rest of the reservoir.

By using the braces and asterisk, it is possible to define the same distribution by means of the following expression.

```
Example
MULTPV
3*{5*200 3*{200 3*1 200} 5*200}
/
```

Below the same distribution is set, using keyword ARITHMETIC (see 6.3.1).

```
Example
ARITHMETIC
MULTPV = 1
MULTPV(1:5, 1:1, 1:3) = 200
MULTPV(1:5, 5:5, 1:3) = 200
MULTPV(1:1, 1:5, 1:3) = 200
MULTPV(5:5, 1:5, 1:3) = 200
/
```

6.2.22   ACTNUM

	tNavigator		eclipse 300		MORE Roxar
√		√			
√	eclipse 100		CMG imex		CMG stars

This keyword specifies active/inactive cells. The number of values should be equal to number of cells. Values should be integer numbers. 1 is for active cell and 0 is for inactive cell.

Default: 1 for all cells

```
Example
DIMENS
5 5 4
/
...
ACTNUM
100*1
/
```

In this example all cells are active cells.

```
Example
DIMENS
5 4 1
/
...
ACTNUM
1 1 1 1 1
1 0 0 0 1
1 0 0 0 1
1 1 1 1 1
/
```

In this example cell $[1, 1, 1]$ is active, but cell $[2, 2, 1]$ is inactive

6.2.23 MINPV

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to switch grid blocks into inactive state if its pore volume is smaller than specified in this keyword.

Default: 0.000001

```
Example
MINPV
100
/
```

In this example all grid blocks with pore volume less than 100 m^3 are switch into inactive state.

6.2.24 MINPVV

✓	**tNavigator**	✓	**eclipse 300**
✓	**eclipse 100**	☐	**CMG imex**

☐ **MORE Roxar**
☐ **CMG stars**

This keyword is used to switch grid blocks into inactive state if its pore volume is smaller than specified in this keyword. The number of values should be equal to number of cells.

Default: 0.000001

```
Example
MINPVV
1 1 1 1 .2 .2 .2 .2 3*10 20
/
```

This example sets different minimal pore volumes for grid blocks. If block pore volume is less that specified number, it will switch into inactive state.

6.2.25 MINDZNET

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to switch grid blocks into inactive state if its net thikness (DZ (see 6.2.1)) is smaller than specified in this keyword (m).

Default: 0.000001

```
Example
MINDZNET
0.2
/
```

In this example all grid blocks with net thickness less than 0.2 m are switch into inactive state.

6.2.26 BNDNUM

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to specify type of boundary condition on vertical boundary surface for all cells. The number of values should be equal to number of cells. Values should be integer numbers. 1 is for Dirichlet boundary condition (constant pressure), 2 is for Neumann boundary condition (non flow). Simulator uses only values in cells located on the boundary. If Dirichlet boundary condition is specified the simulator get pressure from the initial pressure (PRESSURE (see 6.9.7)).

Default: 2 in all cells

```
Example
DIMENS
5 5 4
/
...
BNDNUM
100*2
/
```

In this example Neumann boundary condition is specified in all cells on the boundary.

6.2.27 FAULTS

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

A table is specified which defines a set of faults. Fault transmissibility can later be modified using MULTFLT (see 6.2.28) keyword. Each fault is assigned a name; it can consist of several segments, in this case for all segments this name should be specified. Number of segments for each fault and order of their appearance in fault table are arbitrary. The following data should be specified:

- fault name,

- lower i-coordinate of fault segment,

- upper i-coordinate of fault segment; should be equal to lower i-coordinate if fault face is I or X,

- lower j-coordinate of fault segment,

- upper j-coordinate of fault segment; should be equal to lower j-coordinate if fault face is J or Y,

- lower k-coordinate of fault segment,

- upper k-coordinate of fault segment; should be equal to lower k-coordinate if fault face is K or Z,

- fault face, should be equal to one of the following letters: I, J, K, X, Y, Z.

The table must be terminated with a blank record, containing only a slash (/).

Note: The FAULTS keyword may be used more than once.

Note: The faults defined by means of FAULTS keyword are easy way to modify transmissibility in a group of cells with help of keyword MULTFLT (see 6.2.28). They don't have to coincide with actual faults in the corner-point geometry (see keywords ZCORN and COORD).

```
Example
FAULTS
- NAME   i1 i2 j1 j2 k1 k2 FACE
'fault_1' 8 8   9  20 1 3 'I' /
'fault_2' 8 12 9  9  1 3 'J' /
'fault_2' 8 12 9  20 1 1 'K' /
'fault_1' 8 8  21 21 2 2 'J' /
/
```

This example defines two two-segment faults.

6.2.28 MULTFLT

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword defines transmissibility multipliers for all cells in specified faults, in direction corresponding to face of fault, previously defined in FAULTS (see 6.2.27) keyword. The following data should be specified:

- fault name (as specified in the FAULTS keyword)

- transmissibility multiplier

- diffusivity multiplier, IGNORED (this is an Eclipse compatibility field)

The table must be terminated with a blank record, containing only a slash (/).

Default:

- transmissibility multiplier: 1.0

- diffusivity multiplier: 1.0

Note: If the same fault was specified more than once in MULTFLT keyword, the last multiplier is taken. If both MULTFLT and MULTX (see 6.2.11) or MULTY (see 6.2.13) or MULTZ (see 6.2.15) are used for the same cell face, the effect will be cumulative.

```
Example
FAULTS
- NAME  i1 i2 j1 j2 k1 k2 FACE
'fault_1' 8 8  9  20 1 3 'I' /
/

MULTFLT
- NAME MULT TX MULT D
'fault_1' 0 /
/
```

In this example one fault is defined, transmissibility is set to 0.

```
Example
FAULTS
- NAME  i1 i2 j1 j2 k1 k2 FACE
'fault_1' 8 8  9  20 1 3 'I' /
'fault_2' 8 12 9  9  1 3 'J' /
'fault_2' 8 12 9  20 1 1 'K' /
'fault_1' 8 8  21 21 2 2 'J' /
/

MULTFLT
- NAME MULT TX MULT D
'fault_1' 0.1 /
'fault_1' 0.2 /
'fault_2' 0.3 /
/

MULTFLT
'fault_2' 0.4 /
/

MULTX
1875*10
/
```

In this example two two-segment faults are defined, for fault 1 transmissibility multiplier will be 0.2, for fault 2 0.4 (only the value defined last is used each time). Then MULTX (see 6.2.11) keyword is applied, finally for fault 1, segment oriented in i direction (defined first) transmissibility will be 2. For segment oriented in j direction transmissibility will be 0.2. For fault 2 transmissibility will rest equal to 0.4 (since MULTY and MULTZ are not defined, they are defaulted to 1).

6.2.29 THPRESFT

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword is used to set threshold pressure value for flow through fault. If pressure difference between communicating cells from different fault sides is less than threshold value, there will be no flow between those cells. Otherwise flow between those cells will be calculated basing on pressure difference reduced by threshold value. By default, threshold value is equal to 0.

See also keyword THPRES (see 6.9.6).

Keyword is followed by a 2-column table, each line should end with slash (/). Last table line should contain only slash symbol (/).

The following data should be specified:

- fault name;

- threshold pressure value for flow through this fault.

Default: threshold pressure value = 0.

Note:

- If threshold pressure value for a fault is indicated several times, last value will be used.

- If flow between two gridblocks is governed by THPRES and THPRESFT simultaneously, maximum threshold value will be used.

```
Example
THPRESFT
'fault_1'  10.0 /
'fault_2'  5.0 /
/
```

In this example different threshold pressure values are defined for faults 'fault_1', 'fault_2'.

6.2.30 DUMPFLUX

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

If the keyword is specified a flux file will be written. This file will contain flows across the flux region boundaries. No parameters of the keyword should be entered.

The keyword FLUXNUM (see 6.4.8) for every grid block specifies the flux region to which it belongs.

6.2.31 FLUXREG

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword may be used if there are more than one flux region (FLUXNUM (see 6.4.8)). FLUXREG specifies the active flux regions in the current run. After the keyword one should enter the numbers of active regions. Total number of active regions should be less or equal to the 4-th parameter of the keyword REGDIMS (see 6.1.16). The data should be terminated with a slash /.

The flux regions which are not mentioned as active are considered as inactive, their flows into the active regions are read from the flux file.

The keyword shouldn't be specified in the full-field run (all flux regions are active DUMPFLUX (see 6.2.30)) or if there is only one flux region (the 4th parameter of REGDIMS (see 6.1.16) is equal to 1).

Example
```
FLUXREG
3 5 7 8
/
```

6.2.32   COORD

	tNavigator		eclipse 300		MORE Roxar
√		√		☐	
√	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword is used to specify coordinate lines in Z direction. A coordinate line defines the possible position for grid block corner points, for each (i,j) cell. A coordinate line is specified by two triplets of X, Y and Z coordinates, representing two distinct points on it.

Default: none.

```
Example
DIMENS
1 2 1
/
...
COORD
0 0 1 0 0 2
1 0 1 1 0 2
0 1 1 0 1 2
1 1 1 1 1 2
0 2 1 0 2 2
1 2 1 1 2 2
/
```

In this example for all corner points on the top surface of reservoir specified vertical coordinate lines.

6.2.33   ZCORN

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

Each grid block has 8 corners. This keyword enables the depths of each corner of each grid block to be separately specified. It is used for specifying depths for corner point geometry.

The keyword line is followed by $2*NX*2*NY*2*NZ$ values, with the two corners in the X direction of the first grid block being specified first, then two corners for the next block in the i direction, and so on.

Default: none.

```
Example
DIMENS
1 2 2
/
...
ZCORN
8 * 2500
8 * 2505
8 * 2505
8 * 2515

/
```

In this example top of first layer grid blocks is specified as 2500, DZ for first layer grid blocks is specified as 5 and 10 for DZ in second layer grid blocks.

6.2.34   RESVNUM

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword begins COORD (see 6.2.32) data input for every reservoir. One should enter the number of reservoir for which data will be entered using the keyword COORD (see 6.2.32). The data should be terminated with a slash /.

The number of reservoir should be greater or equal to 1 and less or equal to NUMRES (see 6.1.34). NUMRES (see 6.1.34) should be specified and greater than 1.

```
Example
RESVNUM
1/
COORD
0 0 1   0 0 2
0 1 1   0 1 2
1 0 1   1 0 2
1 1 1   1 1 2
2 0 1   2 0 2
2 1 1   2 1 2
/
RESVNUM
2/
COORD
1 0 1   3 0 2
2 1 1   4 1 2
1 0 1   1 0 2
1 1 1   1 1 2
3 0 1   2 0 2
2 1 1   2 1 2
/
```

6.2.35 COORDSYS

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword provides information about coordinate system for every reservoir. Layers of blocks can be put to several reservoirs at the same time.

The keyword should be specified only if the is more than one reservoir. The keyword should be followed by the NUMRES (see 6.1.34) sets of data (equal to the amount of reservoirs). Each set of data should be terminated with a slash /.
The folowing parameters are to be specified for every set of data:

1. minimal block index number in Z-direction for this reservoir (this number shouldn't be greater than NZ in DIMENS (see 6.1.12));

2. maximal block index number in Z-direction for this reservoir (this number shouldn't be greater than NZ in DIMENS (see 6.1.12));

3. completion of circle for this reservoir (COMP - circle is completed in Y-direction, INCOMP - circle is not completed. The circle can be completed if coincident coordinate lines are identified);

4. connection to the reservoir below (JOIN - transmissibilities to the reservoir below are calculated, SEPARATE - this reservoir is separated with the reservoir below. In JOIN case transmissibilities between blocks (i, j, k) and $(i, j, k+1)$ are calculated (if these blocks belong to different reservoirs). Fault non-neighbor conections are not calculated between different reservoirs);

5. lower bound for reservoir number for lateral block connection between reservoirs;

6. upper bound for reservoir number for lateral block connection between reservoirs. Lower and upper bound may be used to allow fault NNC between cells in different reservoirs. These numbers should lie between 1 and NUMRES (see 6.1.34).

Default:

- completion of circle for this reservoir – INCOMP,

- connection to the reservoir below – SEPARATE,

- lower bound for reservoir number – current reservoir number,

- upper bound for reservoir number – current reservoir number.

```
Example
NUMRES
4
/
COORDSYS
1    4 'INCOMP' 'SEPARATE' 1 1
/
5  16 'INCOMP' 'SEPARATE' 2 2
/
17 20 'INCOMP' 'SEPARATE' 3 3
/
21 54 'INCOMP' 'SEPARATE' 4 4
/
```

6.2.36　GDFILE

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword allows to define model grid by means of file of special format (`*.GRID` format of Eclipse simulator). The grid defined in this file is a set of cubes, with coordinates of all 8 vertexes given, a more flexible way than COORD (see 6.2.32), ZCORN (see 6.2.33) allows. Only UNFORMATTED file format is supported. The following data should be specified (list terminated by a slash /).

- file name (file should be of UNFORMATTED format), with grid geometry

- file format; possible values are: FORMATTED, F, UNFORMATTED, U; if FORMATTED, or F, is indicated, grid will NOT be read from file indicated in previous argument.

Default: file format: UNFORMATTED.

```
Example
GDFILE
filegrid.GRID UNFORMATTED
/
```

In this example grid will be taken from `filegrid.GRID`.

6.2.37 NNC

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

Usually non-neighbor connections are created due to faults, but they can be set explicitly using this keyword. For each non-neighbor connection one line of data is input, with coordinates of cells to be connected and transmissibility between them. Each line should end with slash ($/$). Data table should end with slash, input on the new line.

1. I-coordinate of first cell from non-neighbour connection;

2. J-coordinate of this cell;

3. K-coordinate of this cell;

4. I-coordinate of second cell from non-neighbour connection;

5. J-coordinate of this cell;

6. K-coordinate of this cell;

7. non-neighbor connection transmissibility.

Defautl: non-neighbor connection transmissibility: 0;

Note: In non-neighbor connection is set between cells with nonzero transmissibility, this increases transmissibility between them on this value.

Note: Any non-neighbor connections with transmissibility less than 0.000001 are ignored.

```
Example
NNC
1 2 2 1 5 2 100 /
1 1 2 1 4 2 100 /
/
```

In this example non-neighbor connections between two pairs of grid blocks are set, with transmissibility 100.

6.2.38 EDITNNC

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

Using this keyword one can modify non-neighbor connections generated by faults in the grid. For any non-neighbor connection (you want to modify) you should specify a line of data with following parameters. A line should be terminated by a /. A single / should terminate lines of data.

These parameters are to be specified:

1. I-coordinate of first cell from non-neighbour connection;

2. J-coordinate of this cell;

3. K-coordinate of this cell;

4. I-coordinate of second cell from non-neighbour connection;

5. J-coordinate of this cell;

6. K-coordinate of this cell;

7. $TRANM$ - transmissibility multiplier of this non-neighbor connection; it should be greater or equal to zero; transmissibility is equal to the product of $TRANM$ and non-neighbor connection transmissibility. The last one is calculated by the program or specified by the keyword NNC (see 6.2.37);

8. number of saturation region, corresponding to the flow from the first cell to the second cell;

9. number of saturation region, corresponding to the flow from the second cell to the first cell;

10. number of PVT region, corresponding to the flow from the first cell to the second cell;

11. number of PVT region, corresponding to the flow from the second cell to the first cell;

12. direction, corresponding to the flow from the first cell to the second cell; choose one from this list: $X+, X-, Y+, Y-, Z+, Z-$;

13. direction, corresponding to the flow from the second cell to the first cell; choose one from this list: $X+, X-, Y+, Y-, Z+, Z-$;

14. diffusivity multiplier of this non-neighbor connection (it should be greater or equal to zero); the diffusivity is equal to the product of the diffusivity multiplier and non-neighbor connection diffusivity.

Default:

- *TRANM*: 1.0;

- number of saturation region, corresponding to the flow from the first cell to the second cell (or from the second cell to the first cell): 0;

- number of PVT region, corresponding to the flow from the first cell to the second cell (or from the second cell to the first cell): 0

- diffusivity multiplier: 0.

Example
```
EDITNNC
15 3 1 18 13 1 5 3 1 0 0 Y+ Y- 2.5/
5 28 1 9 33 2 3 1 4 2 1 X+ X- 1.3/
/
```

This example modifies two non-neighbor connections. The transmissibility multiplier of the first non-neighbor connection is set to 5 and directions, corresponding to the flow between cells: $Y+$ $Y-$; the transmissibility multiplier of the second non-neighbor connection is set to 3 and directions, corresponding to the flow between cells: $X+$ $X-$.

6.2.39 TRANX

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword is used to specify transmissibility in X-direction for plane between blocks $[I,J,K]$ and $[I+1,J,K]$. This value replaces value calculated by the program. The keyword should be followed by one non-negative real number for every grid block.

```
Example
DIMENS
5 5 4
/
...
TRANX
100*1.5
/
```

In this example transmissibility is specified for all planes in X-direction as 1.5.

6.2.40 TRANY

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword is used to specify transmissibility in Y-direction for plane between blocks $[I,J,K]$ and $[I,J+1,K]$. This value replaces value calculated by the program. The keyword should be followed by one non-negative real number for every grid block.

```
Example
DIMENS
5 5 4
/
...
TRANY
100*1.5
/
```

In this example transmissibility is specified for all planes in Y-direction as 1.5.

6.2.41 TRANZ

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword is used to specify transmissibility in Z-direction for plane between blocks $[I,J,K]$ and $[I,J,K+1]$. This value replaces value calculated by the program. The keyword should be followed by one non-negative real number for every grid block.

```
Example
DIMENS
5 5 4
/
...
TRANZ
100*1.5
/
```

In this example transmissibility is specified for all planes in Z-direction as 1.5.

6.2.42   PINCH

✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword sets parameters regulating non-neighbor connections for pinched out layers. The following data should be provided, line ending with slash (/).

1. threshold thickness of pinched out layers;

2. control over non-neighbor connections across cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24); possible values – GAP and NOGAP; first one means that non-neighbor connections across cells deactivated by MINPV are created even if thickness exceeds threshold, and second one (NOGAP) – that non-neighbor connections are created always observing threshold thickness;

3. maximal empty gap allowed between cells in adjacent layers for non-zero transmissibility to be allowed between them;

4. method of non-neighbor connection transmissibility calculation through pinched out layers – TOPBOT and ALL; first one means that it will be calculated on base of half-block transmissibilities of active cells below and above pinched-out layer(s), and second one – as harmonic average of cell transmissibilities of pinched out layer(s) between two active cells;

5. method of vertical transmissibility calculation MULTZ (see 6.2.15) across pinched out cells, in case if previous parameter is set to TOPBOT; possible values – TOP and ALL, first one means that value of active cell above pinched out cells will be used, second one – minimum MULTZ (see 6.2.15) of this active cell and all nonactive cells below will be used;

6. threshold pressure difference between cells to make non-neighbour connection active; fluid will flow only if pressure difference exceed this limit.

 Default:

1. threshold thickness of pinched out layers: 0.001;

2. control over non-neighbor connections across cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24): GAP;

3. maximal empty gap allowed between cells in adjacent layers: 1E20;

4. method of non-neighbor connection transmissibility calculation through pinched out layers: TOPBOT;

5. method of vertical transmissibility calculation MULTZ (see 6.2.15) across pinched out cells: TOP;

6. threshold pressure difference: 0.

Example
```
PINCH
0.02 'GAP' /
```

In this example threshold thickness is set to 0.02, and non-neighbor connections through cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24) are set even if thickness exceeds threshold value.

6.2.43 PINCHXY

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify the X and Y-direction pinchout threshold widths.
The data should be terminated by a slash /. Following parameters are to be specified:

1. X-direction threshold width for horizontal pinchout;

2. Y-direction threshold width for horizontal pinchout;

Default

- X-direction threshold width for horizontal pinchout: 0.001;

- Y-direction threshold width for horizontal pinchout: 0.001;

Model's vertical parts which are pinched-out in the X or Y-direction correspond in the grid to columns of inactive cells with zero width in the X or Y-direction. A column of inactive cells doesn't allow a flow (between active cells on each sides of it) to cross it in the X or Y-direction.

In case the keyword PINCHXY is specified, horizontal non-neighbor connections in the X or Y-direction will be created between active cells on each side of the pinch-out column. So fluid will flow through this column. Columns are considered as pinched-out columns in the X or Y-direction if their overall cell width in the X or Y-direction is below the value specified by this keyword.

Non-neighbor connections won't be generated if the keyword NONNC (see 6.1.24) is specified.

```
Example
PINCH
0.02 0.04 /
```

This example sets X-direction threshold width equal to 0.02 and Y-direction threshold width equal to 0.04.

6.2.44   PINCHOUT

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

Analog of PINCH (see 6.2.42), but has no parameters. Equivalent to setting

```
 PINCH
/
```

For more detailed control over non-neighbor connections across pinched out cells see PINCH (see 6.2.42).

Example
```
PINCHOUT
```

In this example non-neighbor connections through pinched out layers will be created, if threshold thickness is not exceeded (for blocks deactivated by MINPV (see 6.2.23) and MINPVV (see 6.2.24) threshold thickness will not be used).

6.2.45   PINCHREG

√	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
√	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword sets parameters regulating non-neighbor connections for pinched out layers for regions identified by PINCHNUM (see 6.2.46). If PINCHNUM is absent, PINCHREG is ignored.

The following data should be provided, one line ending with slash (/) for each pinchout region (data is the same as in PINCH (see 6.2.42)). If the whole line is defaulted, no pinchouts will be generated for this region.

1. threshold thickness of pinched out layers;

2. control over non-neighbor connections across cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24); possible values – GAP and NOGAP; first one means that non-neighbor connections across cells deactivated by MINPV are created even if thickness exceeds threshold, and second one (NOGAP) – that non-neighbor connections are created always observing threshold thickness;

3. maximal empty gap allowed between cells in adjacent layers for non-zero transmissibility to be allowed between them;

4. method of non-neighbor connection transmissibility calculation through pinched out layers – TOPBOT and ALL; first one means that it will be calculated on base of half-block transmissibilities of active cells below and above pinched-out layer(s), and second one – as harmonic average of cell transmissibilities of pinched out layer(s) between two active cells;

5. method of vertical transmissibility calculation MULTZ (see 6.2.15) across pinched out cells, in case if previous parameter is set to TOPBOT; possible values – TOP and ALL, first one means that value of active cell above pinched out cells will be used, second one – minimum MULTZ (see 6.2.15) of this active cell and all nonactive cells below will be used;

6. threshold pressure difference between cells to make non-neighbour connection active; fluid will flow only if pressure difference exceed this limit.

 Default:

1. threshold thickness of pinched out layers: 0.001;

2. control over non-neighbor connections across cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24): GAP;

3. maximal empty gap allowed between cells in adjacent layers: 1E20;

4. method of non-neighbor connection transmissibility calculation through pinched out layers: TOPBOT;

5. method of vertical transmissibility calculation MULTZ (see 6.2.15) across pinched out cells: TOP;

6. threshold pressure difference: 0.

Example
PINCHREG
0.02 'GAP' /
/
0.01 'GAP' /
/

In this example threshold thickness is set to 0.02 in first region, and 0.01 in third one. Non-neighbor connections through cells deactivated using MINPV (see 6.2.23) and MINPVV (see 6.2.24) are set even if thickness exceeds threshold value. No non-neighbor connections will be generated in the second region.

6.2.46 PINCHNUM

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword is used to specify pinchout regions for non-neighborhood connections generation through PINCHREG (see 6.2.45). The number of values should be equal to number of cells. Values should be integer numbers.

Default: 1 in all cells

```
Example
DIMENS
5 5 4
/
...
PINCHNUM
50*1 50*2
/
```

In this example upper half of reservoir belongs to first pinchout region, and lower half of reservoir belongs to second one.

6.2.47 JFUNC

✓	**tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

If end-point scaling is selected by ENDSCALE (see 6.6.11), JFUNC keyword may be specified. In this case capillary pressures will be calculated according to Leverett J-Function option. The following data should be input below. Line should end with a slash (/).

- phase flag: WATER, GAS or BOTH, means what phase capillary pressure will be scaled according to J-function;

- oil-water surface tension, this is ST_W in 2.61, in dynes/cm; if phase flag is equal to WATER, or BOTH, this parameter should be specified;

- oil-gas surface tension, this is ST_G in 2.61, in dynes/cm; if phase flag is equal to GAS, or BOTH, this parameter should be specified;

- power for porosity, this is α in 2.61;

- power for permeability, this is β in 2.61;

- permeability direction: this can be XY, X, Y or Z.

Default:

- power for porosity = 0.5;

- power for permeability = 0.5;

- permeability direction = XY.

Example
```
JFUNC
WATER 22.2 /
```

In this example only oil-water capillary pressure will be scaled, with surface tension equal to 22.2 dynes/cm.

Example
```
JFUNC
GAS 1* 12.2 /
```

In this example only oil-gas capillary pressure will be scaled, with surface tension equal to 12.2 dynes/cm.

Example
```
JFUNC
BOTH 22.2 12.2/
```

In this example both oil-water anf oil-gas capillary pressures will be scaled, with surface tension equal to 22.2 dynes/cm and 12.2 dynes/cm.

6.2.48   MAPAXES

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

To transform grid coordinates to coordinate system associated with the map, MAPAXES may be specified. Usually it is output by grid pre-processors. It is six numbers to be specified:

- X coordinate of one point of the grid Y-axis relative to the map (X1),

- Y coordinate of one point of the grid Y-axis relative to the map (Y1),

- X coordinate of the grid origin relative to the map (X0),

- Y coordinate of the grid origin relative to the map (Y0),

- X coordinate of one point of the grid X-axis relative to the map (X2),

- Y coordinate of one point of the grid X-axis relative to the map (Y2).

Note that the length of vectors (X1, Y1), (X0, Y0) and (X2, Y2), (X0, Y0) should be equal.

Default: nothing

Example
```
MAPAXES
-1 1 0 0 1 1
/
```

In this example the grid is rotated at 45 degrees.

6.2.49 MAPUNITS

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword is used to specify MAPAXES data units. Following units are possible:

- METRES,

- FEET,

- CM.

Default: METRES

Example
MAPUNITS
METRES
/

In this example MAPAXES data units are metres.

6.2.50   LX / LY / LZ

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can be specified in dual porosity run (2.20) if the option viscous displacement is enable (VISCD (see 6.1.40)). The keyword sets the distances between fractures (matrix block sizes) in X, Y and Z directions.

Values are enumerated in X-direction, then — Y-direction, then — Z-direction. The data should be terminated with a slash /.

One should enter NX * NY * (NZ/2) values. The data should be terminated with a slash /.

Default: none. The data will not be used in the calculation of sigma-factor if the keyword LTOSIGMA (see 6.2.55) is enable.

```
Example
DUALPORO
VISCD
...
DIMENS
10 10 2
...
LX
100*800
/
LY
50*550 50*280
/
LZ
100*70
/
```

This example shows the model $NX = 10$, $NY = 10$, $NZ = 2$. Dual porosity is specified using the keyword DUALPORO (see 6.1.37), Viscous displacement is specified using the keyword VISCD (see 6.1.40). The distances between fractures (matrix block sizes) in X-direction is equal to 800 metres for all 100 blocks, in Y-direction for first 50 blocks – 550 metres, for next 50 blocks – 280 metres, in Z-direction for all blocks – 70 metres.

6.2.51   DPNUM

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can be used in dual porosity single permeability run (2.20). The keyword specifies reservoir fields that should be considered as single porosity fields.

One should enter NX * NY * (NZ/2) values. The data should be terminated with a slash /. Dual porosity blocks should be marked by one – 1, single porosity blocks – by zero 0.

Default: If DPNUM isn't specified for the block, this block is considered as dual porosity block.

One should specify grid data for single porosity blocks only in first half of layers in Z direction (NZ/2) (matrix blocks). Fracture blocks (corresponding to single porosity matrix blocks) are inactive.

The keyword will be ignored if DUALPORO (see 6.1.37) isn't specified. DUALPERM (see 6.1.38) shouldn't be specified in the same time with DPNUM, since single porosity is not allowed in dual permeability run.

```
Example
DUALPORO
...
DIMENS
10 10 2
...
DPNUM
10*0 10*1 10*0 10*1
10*0 10*1 10*0 10*1
10*0 10*1
/
```

This example shows the model $NX = 10$, $NY = 10$, $NZ = 2$. Dual porosity single permeability is specified using the keyword DUALPORO (see 6.1.37), then single porosity field is specified (the half of the reservoir).

6.2.52   DPGRID

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can be used in dual porosity single permeability run (2.20). If the keyword is enable one should specify grid data only for matrix blocks (NX * NY * (NZ/2)); values for fracture blocks will be obtained (copied) from corresponding matrix blocks.

This operation is applied for the values specified by following keywords: DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1), PERMX (see 6.2.7), PERMY (see 6.2.7), PERMZ (see 6.2.7), PORO (see 6.2.17), TOPS (see 6.2.5), NTG (see 6.2.18), DZNET (see 6.2.19), ZCORN (see 6.2.33), PERMXY / PERMYZ / PERMXZ (see 6.2.8), DEPTH (see 6.2.6). This operation is applied only for fracture blocks which don't have manually input grid data.

Example
DPGRID

6.2.53 SIGMA

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can be used in dual porosity or dual permeability run (2.20), specified by keywords DUALPORO (see 6.1.37), DUALPERM (see 6.1.38). The keyword sets sigma-factor multiplier that is used in matrix-fracture coupling transmissibilities.

The keyword sets sigma-factor for all blocks. The data should be terminated with a slash /.

Sigma-factor is related to the distances between fractures (matrix block sizes) in X, Y and Z directions:

$$\sigma = 4(\frac{1}{lx^2} + \frac{1}{ly^2} + \frac{1}{lz^2}),$$

lx, ly и lz – the distances between fractures (matrix block sizes) in X, Y and Z directions. (These distances are not the dimensions DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1).)

Default. If no one of the keywords SIGMA, SIGMAV (see 6.2.54), LTOSIGMA (see 6.2.55) is specified, sigma-factor will be considered as zero.

Different sigma-factors for different blocks can be specified using the keyword SIGMAV (see 6.2.54).

Example
```
SIGMA
0.25 /
```

This example sets sigma-factor equal to 0.25.

6.2.54 SIGMAV

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can be used in dual porosity run (2.20), specified by keyword DUALPORO (see 6.1.37). The keyword SIGMAV sets sigma-factor multiplier that is used in matrix-fracture coupling transmissibilities.

One should enter NX * NY * (NZ/2) values. The data should be terminated with a slash /.

Sigma-factor is related to the distances between fractures (matrix block sizes) in X, Y and Z directions:

$$\sigma = 4(\frac{1}{lx^2} + \frac{1}{ly^2} + \frac{1}{lz^2}),$$

lx, ly и lz – the distances between fractures (matrix block sizes) in X, Y and Z directions. (These distances are not the dimensions DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1).)

Default. If no one of the keywords SIGMA (see 6.2.53), SIGMAV, LTOSIGMA (see 6.2.55) is specified, sigma-factor will be considered as zero.

Common sigma-factor for all blocks can be specified using the keyword SIGMA (see 6.2.53).

```
Example
DUALPORO
...
DIMENS
10 10 2
...
SIGMAV
25*0.17 25*1 50*0.26
/
```

This example shows the model $NX = 10$, $NY = 10$, $NZ = 2$. Dual porosity is specified using the keyword DUALPORO (see 6.1.37). First 25 matrix blocks have sigma-factor 0.17, next 25 – 1, next 50 – 0.26.

6.2.55   LTOSIGMA

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can be used in dual porosity run (2.20), if the option Viscous displacement is enable (VISCD (see 6.1.40)). Using the keyword LTOSIGMA sigma-factor multiplier can be obtained from the distances between fractures (matrix block sizes).

The data should be terminated with a slash /. The following parameters are to be specified:

1. fx,

2. fy,

3. fz.

Sigma-factor is related to the distances between fractures (matrix block sizes) in X, Y and Z directions:

$$\sigma = \frac{fx}{lx^2} + \frac{fy}{ly^2} + \frac{fz}{lz^2},$$

lx, ly и lz – the distances between fractures (matrix block sizes) in X, Y and Z directions. (These distances are not the dimensions DX (see 6.2.1), DY (see 6.2.1), DZ (see 6.2.1).) The values of lx, ly, lz that aren't specified or are equal to zero will not be used in calculations.

Default. If no one of the keywords SIGMA (see 6.2.53), SIGMAV (see 6.2.54), LTOSIGMA is specified, sigma-factor will be considered as zero.

Sigma-factor can be specified manually using the keywords SIGMA (see 6.2.53), SIGMAV (see 6.2.54). If LTOSIGMA is enable, manually entered sigma-factor will be ignored.

```
Example
DUALPORO
...
LTOSIGMA
4 4 2
/
```

Dual porosity single permeability is specified using the keyword DUALPORO (see 6.1.37). fx is equal to 4, $fy = 4$, $fz = 2$.

6.2.56 MULTMF

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can be used in dual porosity run (2.20) DUALPORO (see 6.1.37). The keyword specifies multiplier which is used to calculate the matrix-fracture flows.

The same number of values should be specified as the number of matrix blocks. The data should be terminated with a slash /.

6.2.57 ROCKPROP

✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword specifies properties of cap and base rocks, which will be used to model the heat exchange between the reservoir and surroundings. The line contains data for one type of rock. Each line should be terminated with a slash /. The data for all types should be terminated with a final /.

The line of properties for one type of rock contains the following parameters:

1. the number of rock type (this value should be less than the 1-st parameter of the keyword ROCKDIMS (see 6.1.20)),

2. initial temperature (C),

3. rock conductivity ($kJ/m/day/C$),

4. volumetric heat capacity ($kJ/m^3/C$),

5. temperature-dependent coefficient of the volumetric heat capacity of the rock ($kJ/m^3/C^2$),

6. calculation method for heat loss: V - Vinsome and Westerveld, N - numerical (this method is disable in tNavigator).

Default:

- temperature-dependent coefficient of the volumetric heat capacity of the rock – 0,

- calculation method for heat loss – V (Vinsome).

```
Example
ROCKDIMS
2 1* /
...
ROCKPROP
1 65 6 35 2* /
2 70 7 39 2* /
/
```

This example sets properties for two types of cap and base rocks (at first the keyword ROCKDIMS (see 6.1.20) sets the maximum number of rock types – 2).

6.2.58   ROCKCON

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword specifies the connection between the reservoir and cap and base rocks, which will be used to model the heat exchange between the reservoir and surroundings. The properties of cap and base rocks are specified using the keyword ROCKPROP (see 6.2.57). The line contains data for connection between one type of rock and reservoir. Each line should be terminated with a slash /. The data should be terminated with a final /.

Each line contains the following parameters:

1. the number of rock type (this value should be less than the 1-st parameter of the keyword ROCKDIMS (see 6.1.20)),

2. lower number of connectiong grid block in X-direction;

3. upper number of connectiong grid block in X-direction;

4. lower number of connectiong grid block in Y-direction;

5. upper number of connectiong grid block in Y-direction;

6. lower number of connectiong grid block in Z-direction;

7. upper number of connectiong grid block in Z-direction;

8. face of the reservoir to which the rock connects (one of the labels I-, I+ (X-direction), J-, J+ (Y-direction), K- (top face of the reservoir), K+ (bottom face of the reservoir));

9. rock influx coefficient (transmissibility multiplier for the connection between the rock and the reservoir grid).

The number of grid blocks which can be connected to any rock type shouldn't be greater than the 3-rd parameter of the keyword ROCKDIMS (see 6.1.20)).

Default:
The 9-th parameter – 1.

Example
```
ROCKCON
1 1 25 1 25 1 1 K- /
2 1 25 1 25 20 20 K+ /
```

   This example sets the connection of two rock types and the reservoir grid 25x25x20 on top and from the bottom.

6.2.59   ROCKCONT

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword specifies the connection between the reservoir and cap and base rocks, initial temperature, volumetric heat capacity, rock conductivity of reservoir surroundings and minimal difference between temperatures, which will be used to model the heat exchange between the reservoir and surroundings 3.29.

The line contains data for connection between rock region and cap or base rock. Each line should be terminated with a slash /. Different values can be entered for different rock regions (ROCKNUM (see 6.4.6)). The data should be terminated with a final /.

Each line contains the following parameters:

1. rock region number;

2. direction in which heat loss properties are applied: I+, I-, J+, J-, K+, K- (I – X-axis, J – Y-axis, K – Z-axis);

3. initial temperature of reservoir surroundings (C);

4. volumetric heat capacity ($kJ/m^3 - C$);

5. rock conductivity ($kJ/m - day - C$).

6. minimal difference between temperatures when the calculations of the heat exchange should start (C).

The keyword ROCKCONT has Eclipse compatible analogues ROCKCON (see 6.2.58), ROCKPROP (see 6.2.57).

```
Example
ROCKCONT
1 I- 70 35 4 0 /
3 K+ 60 35 4 0 /
/
```

In this example there is the heat exchange between 1-st rock region and reservoir surroundings in I- direction, initial temperature – $70C$, volumetric heat capacity – $35kJ/m^3 - C$, rock conductivity – $4kJ/m - day - C$, minimal difference between temperatures – 0.
The heat exchange between 2-nd rock region and reservoir surroundings in I- direction, initial temperature – $60C$, volumetric heat capacity – $35kJ/m^3 - C$, rock conductivity – $4kJ/m - day - C$, minimal difference between temperatures – 0.

6.2.60 SPECGRID

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword gives the same information as several keywords in the Definition section 6.1. These parameters are not used in the run. They can be used for a checking function.
The data should be terminated with a slash /.
The following parameters are to be specified:

1. the number of blocks in the X-direction (NX in DIMENS (see 6.1.12));

2. the number of blocks in the Y-direction (NY in DIMENS (see 6.1.12));

3. the number of blocks in the Z-direction (NZ in DIMENS (see 6.1.12));

4. the number of reservoirs (NUMRES (see 6.1.34)), there is a coordinate system for each reservoir;

5. type of the coordinate system (T - cylindrical, F - cartesian), tNavigator supports only cartesian coordinates.

```
Example
SPECGRID
10 15 4 2 F
/
```

6.2.61 CARFIN

√ tNavigator	√ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies a cartesian local grid refinement (LGR) – section 2.21. CARFIN specifies a cell or a box of cells identified by its global grid coordinates I1-I2, J1-J2, K1-K2, to be replaced by refined cells. The dimensions of the refined grid within this box are specified as NX, NY, NZ. The data should be terminated with a slash /.

CARFIN can be followed by keywords that describe properties in LGR, if they are different from the properties in parent grid. These keywords should be terminated with the keyword ENDFIN (see 6.2.63), which terminates data for a local grid refinement.

The following parameters should be specified:

1. name of the local grid refinement;

2. I1 Lower coordinate of the box in the parent grid (in X direction);

3. I2 Upper coordinate of the box in the parent grid (in X direction);

4. J1 Lower coordinate of the box in the parent grid (in Y direction);

5. J2 Upper coordinate of the box in the parent grid (in Y direction);

6. K1 Lower coordinate of the box in the parent grid (in Z direction);

7. K2 Upper coordinate of the box in the parent grid (in Z direction);

8. NX Number of refined cells along X direction;

9. NY Number of refined cells along Y direction;

10. NZ Number of refined cells along Z direction;

11. Maximum number of wells this local refined grid will contain;

12. Name of parent LGR. This item may be set to either a null string or to the string GLOBAL to indicate that the parent grid is global (that is, this is not a nested refinement). If the parent grid is already an LGR then the name of the parent LGR should be specified. The range of I, J K indices should then refer to the parent grid. If the parent grid is the global grid, then the range of I, J and K indices refers to the global grid.

Default:

Name of parent LGR – GLOBAL.

Example
```
CARFIN
LGR1 18 18 3 3 1 2 3 1 6 /
PORO
0.22 0.25 0.20 0.22 0.23 0.18
0.16 0.18 0.22 0.16 0.17 0.16
0.21 0.23 0.19 0.12 0.17 0.17 /
ENDFIN
```

In this example there is local grid refinement LGR1 of global grig. Box: layers 1 and 2 of globa l grid (in Z direction), layer 18 in X direction, layer 3 in Y, – are replaced by cartesian LGR 3x1x6 with new values of porosity.

6.2.62 REFINE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword initiates data input for a named local grid (LGR) – section 2.21. The keyword is followed by the local grid name, terminated with a slash /. Subsequent keywords, from the available sets described below, are taken to refer to the specified local grid, until either an ENDFIN (see 6.2.63) (which terminates data for a local grid refinement) or another REFINE (see 6.2.62) keyword is entered.

REFINE keyword can be used in sections GRID, EDIT, PROPS, REGIONS, SOLUTION and SCHEDULE.

GRID section

The REFINE keyword can be used in the GRID section to return to a local grid that has previously been introduced with CARFIN, to make further modifications to its grid data or to supply additional data. In addition, the BOX (see 6.3.2), COPY (see 6.3.7), EQUALS (see 6.3.13), MULTIPLY (see 6.3.4) and COPYBOX (see 6.3.8) keywords can be used to set or modify keywords in regions of the local grid.

EDIT section

The following grid arrays may be edited for a local grid: PORV (see 6.2.20), TRANX (see 6.2.39), DEPTH (see 6.2.6), TRANY (see 6.2.40), TRANZ (see 6.2.41). In addition, the BOX (see 6.3.2), COPY (see 6.3.7), EQUALS (see 6.3.13) keywords can be used to set or modify keywords in regions of the local grid. If any grid block values are defaulted, the host global grid cell value is used.

PROPS section

In end point scaling runs (keyword ENDSCALE (see 6.6.11)) the following keywords and their directional equivalents are available: SWL (see 6.6.14), SWCR (see 6.6.16), SWU (see 6.6.20), SG (see 8.6.8), SGCR (see 6.6.17), SGU (see 6.6.21), SOWCR (see 6.6.18), SOGCR (see 6.6.19), KRO (see 6.6.25), KRORW (see 6.6.25), KRORG (see 6.6.25), KRW (see 6.6.26), KRWR (see 6.6.26), KRG (see 6.6.27), KRGR (see 6.6.27). In addition, the BOX (see 6.3.2), COPY (see 6.3.7), EQUALS (see 6.3.13) keywords can be used to set or modify keywords in regions of the local grid. If any grid block values are defaulted, the host global grid cell value is used.

REGIONS section

The following keywords are available for local grid data input: PVTNUM (see 6.4.1), SATNUM (see 6.4.2), EQLNUM (see 6.4.3), ROCKNUM (see 6.4.6). In addition, the BOX (see 6.3.2), COPY (see 6.3.7), EQUALS (see 6.3.13) keywords can be used to set or modify keywords in regions of the local grid. If any grid block values are defaulted, the

host global grid cell value is used.

SOLUTION section

The following keywords are available for local grid data input: PRESSURE (see 6.9.7), SWAT (see 6.9.9), SGAS (see 6.9.10), RS (see 6.9.19) or PBUB (see 6.9.18), RV (see 6.9.20) or PDEW (see 6.9.21). BOX (see 6.3.2), ADD (see 6.3.11), COPY (see 6.3.7), EQUALS (see 6.3.13) keywords are not available in the SOLUTION section.

SCHEDULE section

The REFINE keyword can be used in the SCHEDULE section to modify region data during a simulation, though modification of SATNUM (see 6.4.2) or PVTNUM (see 6.4.1) data during a simulation should be carried out with care and is not recommended in general. In addition, the BOX (see 6.3.2), COPY (see 6.3.7), EQUALS (see 6.3.13), ADD (see 6.3.11) keywords can be used to set or modify keywords in regions of the local grid. Для ввода данных локальной сетки после ключевого слова REFINE могут использоваться следующие ключевые слова: PVTNUM (see 6.4.1), SATNUM (see 6.4.2), ROCKNUM (see 6.4.6), MULTX (see 6.2.11), MULTY (see 6.2.13), MULTZ (see 6.2.15), MULTPV (see 6.2.21).

In this example in EDIT section for local grid LGR1 the values of pore volume and depth are set in the specified parallelepiped:

```
Example
REFINE
'LGR1' /
EQUALS
PORV 90.0 1 2 1 2 3 4 /
DEPTH 1200.0 1 4 1 4 4 4 /
/
ENDFIN
```

In this example in PROPS section for local grid LGR1 minimal water saturation of cells, used for saturation point scaling, is set – 0.22. Then in the specified parallelepiped value of minimal water saturation is set to 0.20. Then minimal water saturation property array is copied to critical water saturation property array:

```
Example
REFINE
'LGR1' /
SWL
300*0.22 /
EQUALS
SWL 0.20 1 10 1 10 2 2 /
/
COPY
SWL SWCR /
/
ENDFIN
```

In this example in REGIONS section for local grid LGR1 the numbers of saturation function region to which cells belong are set:

```
Example
REFINE
'LGR1' /
SATNUM
35*1
35*2
35*1 /
/
ENDFIN
```

In this example in SOLUTION section for local grid LGR1 the value of initial pressure and initial water saturation is set:

```
Example
REFINE
'LGR1' /
PRESSURE
20*2700
20*2705
20*2710
20*2715 /
SWAT
40*0.21
20*0.25
20*0.27 /
ENDFIN
```

In this example in SCHEDULE section for local grid LGR1 the numbers of saturation function region to which cells belong are set:

```
Example
REFINE
'LGR1' /
SATNUM
10*2
10*1
10*2 /
ENDFIN
```

6.2.63 ENDFIN

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword terminates data for a local grid refinement (LGR) – section 2.21.

CARFIN specifies a cell or a box of cells identified by its global grid coordinates to be replaced by refined cells. CARFIN (or REFINE (see 6.2.62)) can be followed by keywords that describe properties in LGR, if they are different from the properties in parent grid. These keywords should be terminated with the keyword ENDFIN (see 6.2.63), which terminates data for a local grid refinement.

It is not necessary to insert ENDFIN (see 6.2.63) between successive CARFIN (see 6.2.61) or REFINE (see 6.2.62) keywords. The primary purpose of ENDFIN (see 6.2.63) is to revert the program to reading data for the global grid system. In the EDIT, PROPS, REGIONS, SOLUTION and SCHEDULE sections, the ENDFIN (see 6.2.63) keyword tells the program that subsequent data no longer applies to the local grid named in the previous REFINE (see 6.2.62) keyword.

The ENDFIN keyword has no associated data.

6.2.64   NXFIN / NYFIN / NZFIN

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

These keywords can be used to specify number of local cells in each global cell of an LGR (NXFIN – in X direction, NYFIN – in Y direction, NZFIN – in Z direction) (section 2.21).

If a local grid refinement covers more than one global cell in the X direction, NXFIN can be used to specify how many local cells each of the global cells is divided into. The keyword should be placed after the keyword CARFIN (see 6.2.61) introducing the local grid, and before the local grid data is terminated with ENDFIN (see 6.2.63).

NXFIN should be followed by I2-I1+1 values terminated with a slash (/), where I1 and I2 are the I-coordinates defining the box of global grid cells to be refined (parameters 2 and 3 in keyword CARFIN (see 6.2.61)). The number of values is thus the number of global cells of the refinement counted along the X direction. The values represent the number of local cell divisions, counted along the X direction, in each of the global cells. The sum of the values must, of course, be equal to NX set in parameter 8 of CARFIN (see 6.2.61). In the absence of this keyword, the global cells are refined to contain equal numbers of local cells in the X direction.

Analogously for the keywords NYFIN, NZFIN.

Example
```
CARFIN
LGR2 3 4 1 2 5 6 5 2 4 /
NXFIN
3 2 /
```

In this example 2 global cells in X direction are replaced by 5 local cells. First global cell is replaced by 3 local cells, second global cell – by 2 local cells.

6.2.65 HXFIN / HYFIN / HZFIN

√ tNavigator	√ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

These keywords can be used to specify the size ratios of each cell in a local grid refinement (LGR) (HXFIN – in X direction, HYFIN – in Y direction, HZFIN – in Z direction) (section 2.21).

These keywords should be placed after the CARFIN (see 6.2.61) keyword introducing the local grid and before the terminating ENDFIN (see 6.2.63).

HXFIN should be followed by NX values – the total number of cells in the refined grid along the X direction as specified in keyword CARFIN (see 6.2.61) parameter 8. The values represent the X direction size ratios for each the refined grid cells. You may default the size ratios for all the refined cells belonging to a particular host cell. If no ratios are given for a host cell, it is divided up in equal proportions. Each host cell must have the size ratios of its constituent refined cells either all set or all defaulted.

Analogously for the keywords HYFIN, HZFIN.

```
Example
NZFIN
3 2 /
HZFIN
2.0 1.0 3.0
3.0 2.0 /
```

In this example 2 global cells in Z direction are replaced by 5 local cells. In upper (in Z direction) global layer there are 3 local layers (ratios 2:1:3), in bottom global layer – 2 local layers (ratios 3:2).

6.3 Arithmetic section

This section describes all keywords necessary for performing arithmetic operations on grid property arrays. It includes a keyword ARITHMETIC (see 6.3.1), facilitating work with large data arrays and their modification.

6.3.1 ARITHMETIC

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies arithmetic operations on arrays. It is followed by operation description. Arithmetic operations are performed on arrays element by element. For example, PERMX = 2 * PERMY + PERMZ * PORO means $PERMX_{i,j,k} = 2 * PERMY_{i,j,k} + PERMZ_{i,j,k} * PORO_{i,j,k}$. Here equality is an assignment statement, that is: the same array may be used several times in the expression and both in right and left hand sides of the assignment.

The keyword may be present both in initial data section and in schedule section (see below list of keywords with indication whether this array may be modified during calculation – in schedule – by means of ARITHMETICS keyword or not).

Note that scalar value is expanded to an array with all elements equal to this value.

The following binary operations are supported:

- + – element by element summation

- - – element by element subtraction

- * – element by element multiplication

- / – element by element division

- max(·, ·) – element by element maximum calculation

- min(·, ·) – element by element minimum calculation

The following unary operations are supported:

- - – element by element negation

- LOG – element by element logarithm calculation

- EXP – element by element exponent calculation

- SIN – element by element sine calculation

- COS – element by element cosine calculation

- TAN – element by element tangent calculation

- SQRT – element by element square root calculation

- ABS – element by element absolute value calculation

Arrays available in arithmetic:

- Grid:

- DX (see 6.2.1) – size of cells in X-direction
- DY (see 6.2.1) – size of cells in Y-direction
- DZ (see 6.2.1) – size of cells in Z-direction
- PERMX (see 6.2.7) – permeability in X-direction
- PERMY (see 6.2.7) – permeability in X-direction
- PERMZ (see 6.2.7) – permeability in X-direction
- MULTX (see 6.2.11) – transmissibility multipliers in X-direction (also during calculation)
- MULTY (see 6.2.13) – transmissibility multipliers in Y-direction (also during calculation)
- MULTZ (see 6.2.15) – transmissibility multipliers in Z-direction (also during calculation)
- TOPS (see 6.2.5) – depths for top planes of cells in first layer
- NTG (see 6.2.18) – net to gross values
- PORO (see 6.2.17) – porosity values
- PORV (see 6.2.20) – net pore volume
- MULTPV (see 6.2.21) – pore volume multipler

- Initial conditions:

  - SWAT (see 6.9.9) – initial water saturation
  - SGAS (see 6.9.10) – initial gas saturation
  - PRESSURE (see 6.9.7) – initial pressure
  - PBUB (see 6.9.18) – initial bubble point pressure
  - RS (see 6.9.19) – initial gas solution in oil
  - RV (see 6.9.20) – initial oil vaporization in gas
  - PDEW (see 6.9.21) – initial dew point pressure

- Regions:

  - ACTNUM (see 6.2.22) – active/inactive cells
  - BNDNUM (see 6.2.26) – boundary condition
  - EQLNUM (see 6.4.3) – equilibrium regions
  - FIPNUM (see 6.4.4) – FIP regions
  - SATNUM (see 6.4.2) – Saturation regions (also during calculation)
  - PVTNUM (see 6.4.1) – PVT regions (also during calculation)

- End-point scaling:

  – SWL (see 6.6.14) – minimal water saturation
  – SWCR (see 6.6.16) – critical water saturation
  – SWU (see 6.6.20) – maximal water saturation
  – SOWCR (see 6.6.18) – critical oil-to-water saturation
  – PCW (see 6.6.28) – water capillary pressure
  – SWATINIT (see 6.6.30) – initial water saturation
  – SGL (see 6.6.15) – minimal gas saturation
  – SGCR (see 6.6.17) – critical gas saturation
  – SGU (see 6.6.21) – maximal gas saturation
  – SOGCR (see 6.6.19) – critical oil-to-gas saturation
  – PCG (see 6.6.29) – gas capillary pressure

- Grid dimensions (only in the right hand side of arithmetic expressions):

  – NX (see 6.1.12) – number of blocks in X direction
  – NY (see 6.1.12) – number of blocks in Y direction
  – NZ (see 6.1.12) – number of blocks in Z direction

Usual operation priorities are used. Operations may be grouped by parentheses forming more complex expressions like PERMX = LOG (X) * max (TOPS, 2000) * (2 * PERMY + PERMZ * PORO).

For performing operation on a subarray, subarray scopes should be specified in brackets: PERMX(i1:i2,j1:j2,k1:k2) = PERMY + PERMZ * PORO. All the arrays in the right-hand side of the formula will be taken in these scopes.

```
Example
ARITHMETIC
PERMX = 2 * PERMY + PERMZ * PORO
/
```

This example sets permeability in X direction equal to sum of doubled permeability in Y direction and product of permeability in Z direction and porosity. This operation is performed on all grid blocks.

The next example illustrates performing arithmetic operation on an array subregion:

```
Example
ARITHMETIC
PERMX(50:100,10:20,2:2) = 2 *PERMX + PERMY + PERMZ * PORO
/
```

Here porosity and X, Y, Z permeability values, defined above, are used to redefine permeability in X direction for the following subarray: in X direction blocks from 50th to 100th are taken, in Y direction — from 10th to 20th, and in Z direction — layer 2.

6.3.2 BOX

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to define current input box to edit grid properties. Six numbers must be specified. (For example: IMIN IMAX JMIN JMAX KMIN KMAX).

Example
```
DIMENS
10 10 50
/
...
BOX
5 10 5 10 20 30
/
```

This example specifies a 5x5x10 box in a 10x10x50 grid.

6.3.3 ENDBOX

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword has no parameters, it reset the current input box to the entire grid.

```
Example
DIMENS
10 10 50
/
...
BOX
5 10 5 10 20 30
/
...
ENDBOX
/
```

In this example there were specified a 5x5x10 box, and after applying **ENDBOX** the current input box encompasses the entire grid.

6.3.4   MULTIPLY

☑	**tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to multiply grid properties by a constant in current box. It should be followed by grid parameter to be modified, non-negative constant and current box. Box is specified similarly to BOX (see 6.3.2). If it is not specified than last defined box will be used.

Each line should be terminated with a slash /. The data should be terminated with a final slash /.

```
Example
DIMENS
10 10 50
/
...
MULTIPLY
PERMX 1.5   5 10 5 10 20 30 /
PERMY 2.0 /
PERMZ 10 /
/
```

In this example multipliers for parameters PERMX, PERMY, PERMZ of cells in 5x5x10 box are set. Box is specified once.

6.3.5 MULTIREG

√ **tNavigator**	√ **eclipse 300**	☐ **MORE Roxar**
√ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to multiply grid properties by a constant in a specific region. The regions must previously have been defined using keyword MULTNUM (see 6.4.12), OPERNUM (see 6.4.11) or FLUXNUM (see 6.4.8).

An arbitrary number of lines could be specified. Each line should be terminated with a slash /. The data should be terminated with a final slash /.

Each line consists of the following data:

1. the name of the array to be modified;

2. the constant by which the array, specified by parameter 1, is to be multiplied (non-negative);

3. the region number (MULTNUM (see 6.4.12), OPERNUM (see 6.4.11) or FLUXNUM (see 6.4.8)) (region is specified via parameter 4);

4. region M (MULTNUM (see 6.4.12)), O (OPERNUM (see 6.4.11)), F (FLUXNUM (see 6.4.8)).

Default: region – M.

```
Example
DIMENS
5 3 2
/
...
MULTNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
...
MULTIREG
PERMX 1.5 1 /
PERMX 2   2 /
PERMX 10  3 /
PERMX 0.7 4 /
/
```

In this example multiplier for parameter PERMX is set for each of four regions.

6.3.6 MULTREGP

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies pore volume multiplier for a specific region. The regions must previously have been defined using keyword MULTNUM (see 6.4.12) or FLUXNUM (see 6.4.8).

The keyword should be followed by region number and a multiplier.

Each line should be terminated with a slash /. The data should be terminated with a final slash /.

```
Example
DIMENS
5 3 2
/
...
MULTNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
...
MULTREGP
1 1.5 /
2 2.0 /
3 0.0 /
4 0.5 /
/
```

In this example pore volume multiplier is set for each of four regions.

6.3.7 COPY

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword is used to copy data from one property array to another in current box. It should be followed by property array from which data is to be copied, than property array to which data is to be copied and input box. Box is specified similarly to BOX (see 6.3.2).

If it is not specified than last defined box will be used. There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
BOX
5 10 5 10 20 30
/
...
COPY
MULTX MULTY /
MULTX MULTZ /
/
```

In this example values MULTY and MULTZ are set to MULTX in each cell in 5x5x10 box.

6.3.8 COPYBOX

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to copy grid property data from one input box to another. It should be followed by property array to be modified, box from which data is to be copied, than box to which data is to be copied. Box is specified similarly to BOX (see 6.3.2).

There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
COPYBOX
MULTX 1 5 1 5 10 20   6 10 6 10 10 20 /
DY    2 4 1 5 1  10   2 4  1 5  10 20 /
/
```

In this example MULTX and DY data is copied from one input box to another.

6.3.9   COPYREG

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to copy data from one property array to another in a specific region. The regions must previously have been defined using keyword MULTNUM (see 6.4.12), OPERNUM (see 6.4.11) or FLUXNUM (see 6.4.8). The data should be terminated with a slash /.

The following parameters should be specified:

1. property array from which data is to be copied;

2. property array to which data is to be copied;

3. region number (the possible regions are specified via keywords - MULTNUM (see 6.4.12), OPERNUM (see 6.4.11), FLUXNUM (see 6.4.8));

4. region (M – MULTNUM (see 6.4.12), O – OPERNUM (see 6.4.11), F – FLUXNUM (see 6.4.8)).

Default: region – MULTNUM (see 6.4.12).

```
Example
DIMENS
5 3 2
/
...
MULTNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
...
COPYREG
MULTX MULTY 2
/
```

In this example MULTY data is set to MULTX one in each cell of second region.

6.3.10 EQUALREG

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to set property array to a constant value in a specific region. The regions must previously have been defined using keyword MULTNUM (see 6.4.12), OPERNUM (see 6.4.11) or FLUXNUM (see 6.4.8). The data should be terminated with a slash /.

The following parameters should be specified:

1. property array to which constant value is to be set;

2. constant value;

3. region number (the possible regions are specified via keywords - MULTNUM (see 6.4.12), OPERNUM (see 6.4.11), FLUXNUM (see 6.4.8));

4. region (M – MULTNUM (see 6.4.12), O – OPERNUM (see 6.4.11), F – FLUXNUM (see 6.4.8)).

Default: region – MULTNUM (see 6.4.12).

```
Example
DIMENS
5 3 2
/
...
FLUXNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
...
COPYREG
PORO 0.15 1 F/
PORO 0.12 2 F/
PORO 0.08 3 F/
PORO 0.25 4 F/
/
```

In this example porosity is assigned to different constant values in four regions FLUXNUM (see 6.4.8) (0.15 – in 1-st region, 0.12 – in 2-nd region, 0.08 – in 3-rd region, 0.25 – in 4-th region).

6.3.11   ADD

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

   This keyword is used to add a constant to grid property array in current box. It should be followed by grid parameter to be modified, a constant and current box. Box is specified similarly to BOX (see 6.3.2). If it is not specified than last defined box will be used.

   There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
ADD
PERMX 10.5   5 10 5 10 20 30 /
PERMY 20.3 /
PERMZ 10   /
/
```

   In this example adding constats for parameters PERMX, PERMY, PERMZ of cells in 5x5x10 box are set.

6.3.12 ADDREG

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to add a constant to grid property array in a specific region. The regions must previously have been defined using keyword MULTNUM (see 6.4.12), OPERNUM (see 6.4.11) or FLUXNUM (see 6.4.8). The keyword should be followed by grid parameter to be modified, a constant and a region number.

There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
4 3 2
/
...
FLUXNUM
2 2 2 2 2 1 1 1 1 1 3 3
2 2 2 2 2 1 1 1 1 1 3 3
/
...
ADDREG
PERMX 10.5 1 /
SATNUM 1    3 /
/
```

In this example a constant 10.5 is added to PERMX data in each cell of first region, and a constant 1 is added to SATNUM data in each cell of third region.

6.3.13 EQUALS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to set grid property array in current box to a constant. It should be followed by grid parameter to be modified, a constant and current box. Box is specified similarly to BOX (see 6.3.2). If it is not specified than last defined box will be used.

There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
EQUALS
DX 100    5 10 5 10 20 30 /
DY 100 /
DZ 250 /
/
```

In this example for each cell in 5x5x10 box dimentions are set to 100x100x250.

6.3.14 MAXVALUE

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to set a maximum limit to grid property array in current box. It should be followed by grid parameter to be modified, a maximum value and current box. Box is specified similarly to BOX (see 6.3.2). If it is not specified than last defined box will be used.

There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
MAXVALUE
SGU 0.85    5 10 5 10 20 30 /
SWU 0.75 /
/
```

In this example for each cell in 5x5x10 box maximum limit of gas saturation is set to 0.85 and maximum limit of water saturation is set to 0.75.

6.3.15   MINVALUE

	tNavigator		eclipse 300		MORE Roxar
√		√		☐	
√	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword is used to set a minimum limit to grid property array in current box. It should be followed by grid parameter to be modified, a maximum value and current box. Box is specified similarly to BOX (see 6.3.2). If it is not specified than last defined box will be used.

There may be any number of records, each of which is terminated by a slash /.

```
Example
DIMENS
10 10 50
/
...
MINVALUE
SGU 0.45    5 10 5 10 20 30 /
SWU 0.55 /
/
```

In this example for each cell in 5x5x10 box minimum limit of gas saturation is set to 0.45 and minimum limit of water saturation is set to 0.55.

6.3.16   OPERATE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to specify arithmetic operations on grid property arrays in current box. Input parameters:

- grid property array to be modified,

- input box to be modified, it is specified similarly to BOX (see 6.3.2),

- operation, possible operations are represented in table below:

Result	Operation	Argument	First parameter	Second parameter
$R_i = aX_i + b$	MULTA	X	a	b
$R_i = R_i + aX_i^b$	POLY	X	a	b
$R_i = 10^{(a+bX_i)}$	SLOG	X	a	b
$R_i = \log 10(X_i)$	LOG10	X	-	-
$R_i = \ln(X_i)$	LOGE	X	-	-
$R_i = \frac{1}{X_i}$	INV	X	-	-
$R_i = aX_i$	MULTX	X	a	-
$R_i = a + X_i$	ADDX	X	a	-
$R_i = X_i$	COPY	X	-	-
$R_i = \max(a, X_i)$	MAXLIM	X	a	-
$R_i = \min(a, X_i)$	MINLIM	X	a	-
$R_i = aX_i^b$	MULTP	X	a	b

- grid property array as argument,

- first scalar parameter, if required,

- second scalar parameter, if required.

```
Example
DIMENS
10 10 50
/
...
OPERATE
MULTY 4 5 4 5 1 15 COPY MULTX
/
```

In this example MULTX data is copied to MULTY one in each cell in 2x2x15 box.

6.3.17 OPERATER

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to specify arithmetic operations on grid property arrays in a specific region. The regions must previously have been defined using keyword OPERNUM (see 6.4.11).

Input parameters:

- grid property array to be modified,

- input box to be modified, it is specified similarly to BOX (see 6.3.2),

- operation, possible operations are represented in table below:

Result	Operation	Argument	First parameter	Second parameter
$R_i = aX_i + b$	MULTA	X	a	b
$R_i = R_i + aX_i^b$	POLY	X	a	b
$R_i = 10^{(a+bX_i)}$	SLOG	X	a	b
$R_i = \log 10(X_i)$	LOG10	X	-	-
$R_i = \ln(X_i)$	LOGE	X	-	-
$R_i = \frac{1}{X_i}$	INV	X	-	-
$R_i = aX_i$	MULTX	X	a	-
$R_i = a + X_i$	ADDX	X	a	-
$R_i = X_i$	COPY	X	-	-
$R_i = \max(a, X_i)$	MAXLIM	X	a	-
$R_i = \min(a, X_i)$	MINLIM	X	a	-
$R_i = aX_i^b$	MULTP	X	a	b

- grid property array as argument,

- first scalar parameter, if required,

- second scalar parameter, if required.

Example
```
DIMENS
5 3 1
/
...
OPERNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
...
OPERATER
MULTY 2 COPY MULTX
/
```

In this example MULTX data is copied to MULTY one in each cell of second region.

6.4 Region section

This section allows to define regions with different properties. It is also possible to define regions for reserves calculation and regions with different initial conditions (EQUIL (see 6.9.1)).

Region numbers cannon exceed those ones defined by TABDIMS (see 6.1.13) and EQLDIMS (see 6.1.14).

6.4.1 PVTNUM

√ tNavigator	√ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword should be followed by one integer for every grid block specifying the PVT region to which it belongs. The region number should not be less than 1 or greater than the second number specified in keyword TABDIMS (see 6.1.13).

For each PVT region all its PVT properties must be specified, see keywords PVDO (see 6.5.1), PVTO (see 6.5.3), PVTW (see 6.5.4), PVDG (see 6.5.6), PVTG (see 6.5.7), ROCK (see 6.5.12), DENSITY (see 6.5.15), refRFDSALTTRM, SALTPROP (see 6.5.26).

Default: 1

```
Example
DIMENS
5 3 2
/
...
TABDIMS
2 3 2* 4
/
...
PVTNUM
1 1 1 1 1 2 2 2 2 2 3 2 3 3 3
1 1 1 1 1 2 2 2 2 2 3 2 3 3 3
/
```

This example defines disposition of three PVT-regions with different PVT properties for a 5x3x2 grid. Note that TABDIMS specifies that there can be 3 regions with different PVT properties.

```
Example
DIMENS
5 3 2
/
...
TABDIMS
2 3 2* 4
/
...
PVTNUM
2*{5*1 5*2 3 2 3 3 3}
/
```

This example is equivalent to the previous one, but it uses short form with braces.

6.4.2 SATNUM

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword should be followed by one integer for every grid block specifying the saturation function region to which it belongs. The region number should not be less than 1 or greater than the first number specified in TABDIMS (see 6.1.13).

For each saturation function region its relative permeability curves should be specified, see keywords SWOF (see 6.6.1), SGOF (see 6.6.2).

Default: 1

```
Example
DIMENS
5 3 2
/
...
TABDIMS
2 3 2* 4
/
...
SATNUM
2 2 2 2 2 1 1 1 1 1 2 2 1 1 1
2 2 2 2 2 1 1 1 1 1 2 2 1 1 1
/
```

This example defines disposition of two flow regions with different PVT properties for a 5x3x2 grid. Note that TABDIMS specifies that there can be 2 regions with different flow parameters (relative permeabilities).

```
Example
DIMENS
5 3 2
/
...
TABDIMS
2 3 2* 4
/
...
SATNUM
2*{5*2 5*1 2 2 1 1 1}
/
```

This example is equivalent to the previous one, but it uses short form with braces.

6.4.3 EQLNUM

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword should be followed by one integer for every grid block specifying the equilibrium region to which it belongs. The region number should not be less than 1 or greater than the third number specified in EQLDIMS (see 6.1.14).

For each equilibrium region its initial conditions must be specified (EQUIL (see 6.9.1)).

Note: For each equilibrium region grid blocks from this region should not have different numbers of PVT regions (PVTNUM (see 6.4.1)) and should not have different numbers of saturation regions (SATNUM (see 6.4.2)).

Default: 1

```
Example
DIMENS
5 3 2
/
...
EQLDIMS
2
/
...
EQLNUM
2 2 2 2 2 1 1 1 1 1 2 2 1 1 1
2 2 2 2 2 1 1 1 1 1 2 2 1 1 1
/
```

This example defines disposition of two equilibration regions for a 5x3x2 grid. Note that EQLDIMS specifies that there can be 2 equilibration regions.

```
Example
DIMENS
5 3 2
/
...
EQLDIMS
2
/
...
EQLNUM
2*{5*2 5*1 2 2 1 1 1}
/
```

This example is equivalent to the previous one, but it uses short form with braces.

6.4.4 FIPNUM

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword should be followed by one integer for every grid block specifying the fluid-in-place region to which it belongs. The region number should not be less than 1 or greater than the fifth number specified in TABDIMS (see 6.1.13).

For each fluid-in-place region summary information on reserves is calculated and is written down to log file unless other requirements are specified in REPORTFILE (see 6.1.2).

Default: 1

```
Example
DIMENS
5 3 2
/
...
TABDIMS
3 2 2* 4
/
...
FIPNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four fluid-in-place regions for a 5x3x2 grid. Note that TABDIMS specifies that it is possible to define up to 4 fluid-in-place regions.

```
Example
DIMENS
5 3 2
/
...
TABDIMS
3 2 2* 4
/
...
FIPNUM
2*{5*2 5*1 3 3 4 4 4}
/
```

This example is equivalent to the previous one, but it uses short form with braces.

6.4.5 FIP

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies additional sets of fluid-in-place regions. (One can create sets of fluid-in-place regions different from standard FIPNUM (see 6.4.4) regions).

The name of this keyword should contain two parts:

- FIP

- 1-3 symbols (english letters and numbers can be used)

This sequence of symbols specifies the name of one set of fluid-in-place regions.

The keyword should be followed by one integer for every grid block specifying the fluid-in-place region to which it belongs. The number of given integers should be equal to the number of blocks.

You can specify an arbitrary number of sets of fluid-in-place regions (but not greater than the second parameter of the keyword REGDIMS (see 6.1.16)).

For each fluid-in-place region summary information on reserves is calculated and is written down to log file unless other requirements are specified in REPORTFILE (see 6.1.2).

```
Example
DIMENS
3 3 2
/
...
REGDIMS
3 2
/
...
FIP1
1 1 1 1 2 2 3 3 3 1 1 1 1 2 2 3 3 3
/
```

This example defines a set FIP1 of three fluid-in-place regions for 3x3x2 grid. Note that REGDIMS specifies that it is possible to define up to 2 sets of fluid-in-place regions (2-nd parameter of REGDIMS), the number of fluid-in-place regions in one set can be up to 3 (1-th parameter of REGDIMS).

Example
```
DIMENS
25 25 3
/
...
REGDIMS
3 3
/
...
FIPMA1
1000*1 400*2 475*3
/
FIPMA2
875*1 1000*2
/
```

This example defines two sets FIPMA1 and FIPMA2 of fluid-in-place regions for 25x25x3 grid. FIPMA1 consists of three fluid-in-place regions, FIPMA2 — of two.

Note that REGDIMS specifies that it is possible to define up to 3 sets of fluid-in-place regions (2-nd parameter of REGDIMS), the number of fluid-in-place regions in one set can be up to 3 (1-th parameter of REGDIMS).

6.4.6 ROCKNUM

	tNavigator	√	eclipse 300	☐	MORE Roxar
√	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword should be followed by one integer for every grid block specifying the number of rock region - transmissibility dependence on pressure table - to which it belongs. (Tables for each rock region are specifying using the keyword ROCKTAB (see 6.2.10)).

The table number should not be less or equal the second parameter of ROCKCOMP (see 6.1.19).

```
Example
DIMENS
5 3 2
/
...
ROCKCOMP
1 4 1
/
...
ROCKNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four rock regions for 5x3x2 grid. Note that ROCKNUM specifies that it is possible to define up to 4 rock regions.

6.4.7 ENDNUM

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword should be followed by one integer for every grid block, specifying the end point scaling versus depth table region to which it belongs. The region number specifies which end point scaling versus depth table (input using ENPTVD (see 6.6.22), ENKRVD (see 6.6.23)) should be used to calculate the saturation table end points for each grid block.

Table number should not be greater than 3-rd parameter of the keyword ENDSCALE (see 6.6.11). The end point scaling option should be activated by specifying keyword ENDSCALE (see 6.6.11). The same number of numbers should be entered as the number of grid blocks. The data should be terminated with a slash /.

```
Example
DIMENS
12 4 5
/
...
ENDSCALE
2* 2 1*
/
...
ENDNUM
144*1 96*2/
```

In this example (the grid 12x4x5) for all grid blocks the end point scaling versus depth table region is specified: 1 – 144 grid blocks, 2 – next 96 grid blocks.

6.4.8 FLUXNUM

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword should be followed by one integer for every grid block specifying the flux region to which it belongs. The region number should not be less than 1 or greater than the forth number specified in REGDIMS (see 6.1.16).

For each flux region grid property array operations may be specified further, see keywords ADDREG (see 6.3.12), COPYREG (see 6.3.9), MULTIREG (see 6.3.5), MULTREGP (see 6.3.6).

Default: 1

```
Example
DIMENS
5 3 2
/
...
REGDIMS
2 2 0 4
/
...

FLUXNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four flux regions for a 5x3x2 grid. Note that REGDIMS specifies that it is possible to define up to 4 flux regions.

6.4.9 VISCNUM

	tNavigator		eclipse 300		MORE Roxar
√		☐		☐	
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword should be followed by one integer for every grid block specifying the viscosity region number to which it belongs. The region number should not be greater than the second parameter of TABDIMS (see 6.1.13). (The maximum number of viscosity regions is equil to the maximum number of PVT regions).

Default:

In stars data format all blocks belong to the same viscosity region.

In e300 data format viscosity regions coincide with PVT regions (PVTNUM (see 6.4.1)).

```
Example
DIMENS
5 3 2
/
...
TABDIMS
3 4 2*
/
...
VISCNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example specifies four viscosity regions for the grid 5x3x2. Note that TABDIMS specifies that it is possible to define up to 4 viscosity regions.

6.4.10   EOSNUM

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword should be followed by one integer for every grid block specifying the equation of state region (EoS region) to which it belongs. The region number should not be less than 1 or greater than the 9-th parameter of TABDIMS (see 6.1.13). The data should be terminated with a slash /.

Default: All grid blocks belong to the same region.

```
Example
DIMENS
5 3 2
/
...
EOSNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four equation of state regions for a 5x3x2 grid.

6.4.11   OPERNUM

	tNavigator		eclipse 300		MORE Roxar
√		√		☐	
√	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword should be followed by one integer for every grid block specifying the performing arithmetic operations region to which it belongs.

For each performing arithmetic region grid property array operations may be specified further with keyword OPERATER (6.3.17).

Default: 1

```
Example
DIMENS
5 3 2
/
...
OPERNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four performing arithmetic operations regions for a 5x3x2 grid.

6.4.12 MULTNUM

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword should be followed by one integer for every grid block specifying the region of applying multipliers to which it belongs.

For each region of applying multipliers pore volume multipliers may be specified further with keyword MULTREGP (see 6.3.6).

Default: 1

```
Example
DIMENS
5 3 2
/
...
MULTNUM
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
2 2 2 2 2 1 1 1 1 1 3 3 4 4 4
/
```

This example defines disposition of four regions of applying multipliers for a 5x3x2 grid.

6.5   Property section

This section describes reservoir fluid properties as well as properties of reservoir itself. The keywords below allow to describe different two-phase and three-phase models (in particular three-component isothermic black-oil model).

The following keywords set PVT properties of water, oil and gas phases:

- PVTO (see 6.5.3) — live oil;

- PVCO (see   6.5.5) — live oil (The difference between PVCO and PVTO (see 6.5.3) is following: PVCO is used if undersaturated oil has a pressure-independent compressibility and the derivative of viscosity of undersaturated oil is pressure-independent too);

- PVDO (see 6.5.1) — dead oil;

- PVCDO (see 6.5.2) — dead oil (with constant compressibility);

- RSCONST (see 6.5.9) — if dead oil is used to model oil with a constant dissolved gas concentration;

- RSCONSTT (see 6.5.10) — if dead oil is used to model oil with a constant dissolved gas concentration (dissolved gas concentration may be different in different PVT regions);

- PVTG (see 6.5.7) — wet gas;

- PVDG (see 6.5.6) — dry gas;

- PVZG (see 6.5.8) — dry gas (with compressibility factor — Z-factor);

- RVCONST (see 6.5.11) — if dry gas is used to model gas with a constant vaporized oil concentration;

- PVTW (see 6.5.4) — water.

The keywords specifying initial pressures and saturations (see Initialization section (6.9)):

- RV (see 6.9.20) — initial oil vaporization with wet gas (wet gas);

- PDEW (see 6.9.21) — initial dew point pressure (wet gas);

- RS (see 6.9.19) — initial gas solution in oil (live oil);

- PBUB (see 6.9.18) — initial bubble point pressure (live oil).

There are two keyword groups (1 — wet gas, 2 — live oil). One should use only one keyword from one group.

group 1: RV (see 6.9.20), PDEW (see 6.9.21)

group 2: RS (see 6.9.19), PBUB (see 6.9.18)

The keywords specifying relative permeabilities (see Relative permeabilities and capillary pressures (6.6)):

- SWOF (see 6.6.1) — oil-in-water relative permeability (both oil and water phases should be present);

- SGOF (see 6.6.2) — oil-in-gas relative permeability (both oil and gas phases should be present);

- SWFN (see 6.6.4) — water saturation function;

- SGFN (see 6.6.5) — gas saturation function;

- SOF2 (see 6.6.3) — oil saturation function (only for two-phase models);

- SOF3 (see 6.6.6) — oil saturation function (only for three-phase models);

- SGWFN (see 6.6.7) — gas-water saturation function (may be used in gas-water models, when oil isn't present).

The are two keyword groups; one shouldn't mix keywords from different groups.

group 1: SWOF (see 6.6.1), SGOF (see 6.6.2)

group 2: SWFN (see 6.6.4), SGFN (see 6.6.5), SOF2 (see 6.6.3), SGWFN (see 6.6.7), SOF3 (see 6.6.6),

The following keyword combinations are allowed (from keywords PVTO, PVCO, PVCDO, PVDO, PVDG, PVZG, PVTG, PVTW, RSCONST, RSCONSTT, RVCONST, PBUB, RS, PDEW, RV, SWOF, SGOF, SOF2, SWFN, SGFN, SGWFN) :

Live oil, wet gas, water	PVTO (see 6.5.3) (or PVCO (see 6.5.5)), PVTG (see 6.5.7), PVTW (see 6.5.4), SGOF (see 6.6.2), SWOF (see 6.6.1)
Live oil, dry gas, water	PVTO (see 6.5.3) (or PVCO (see 6.5.5)), PVDG (see 6.5.6) (or PVZG (see 6.5.8)), PVTW (see 6.5.4), SGOF (see 6.6.2), SWOF (see 6.6.1)
Live oil, wet gas	PVTO (see 6.5.3) (or PVCO (see 6.5.5)), PVTG (see 6.5.7), SGOF (see 6.6.2), (or SOF2 (see 6.6.3) + SGFN (see 6.6.5))
Undersaturated oil and constant dissolved gas water	PVDO (see 6.5.1) (or PVCDO (see 6.5.2)), RSCONST (see 6.5.9) (or RSCONSTT (see 6.5.10)), PVTW (see 6.5.4), SWOF (see 6.6.1), (or SOF2 (see 6.6.3) + SWFN (see 6.6.4))
Dry gas and constant vaporised oil, water	PVDG (see 6.5.6) (or PVZG (see 6.5.8)), RVCONST (see 6.5.11), PVTW (see 6.5.4), SGFN (see 6.6.5), SWFN (see 6.6.4), (or SGWFN (see 6.6.7))
Dead oil Water	PVDO (see 6.5.1) (or PVCDO (see 6.5.2)), PVTW (see 6.5.4), SWOF (see 6.6.1), (or SOF2 (see 6.6.3) + SWFN (see 6.6.4))
Dry gas, water	PVDG (see 6.5.6) (or PVZG (see 6.5.8)), PVTW (see 6.5.4), SGWFN (see 6.6.7)
Dead oil, Wet gas, water	PVDO (see 6.5.1) (or PVCDO (see 6.5.2)), PVTG (see 6.5.7), PVTW (see 6.5.4), SGOF (see 6.6.2), SWOF (see 6.6.1)

6.5.1 PVDO

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets black oil PVT properties for all PVT regions. The following parameters are to be specified:

- bubble point pressure (p_O argument in 2.9, 2.13, 2.8),

- oil formation volume factor (B_O in 2.9),

- oil viscosity at bubble point (μ_O in 2.8).

For each PVT region oil PVT properties tables should follow, each table must be terminated by a slash /. There must be the same number of tables as number of PVT regions defined in TABDIMS (see 6.1.13).

Default: none

```
Example
TABDIMS
1 2 2* 1
/
...
PVDO
  83.20  1.15  2.45
 239.00  1.12  2.93
 /
 102.34  1.15  2.56
 267.67  1.11  2.89
 /
```

This example sets black oil PVT properties for two PVT regions. Note that TABDIMS (see 6.1.13) keyword defined 2 PVT regions.

6.5.2 PVCDO

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets dead oil (with constant compressibility) PVT properties for all PVT regions. The following parameters are to be specified:

1. reference pressure (p_{ref} in 2.9, 2.8),

2. oil formation volume factor at the reference pressure (B_O in 2.9),

3. oil compressibility $c_O = -\dfrac{\partial B_O}{\partial P} \dfrac{1}{B_O}$,

4. oil viscosity at the reference pressure (μ_O in 2.8),

5. oil viscosibility ($C_v = \dfrac{\partial \mu_O}{\partial P} \dfrac{1}{\mu_o}$).

Use new line to enter oil PVT properties of any region. Each line should be terminated by a slash /. The number of lines should be equal to the number of PVT regions entered by the keyword TABDIMS (6.1.13).

The oil formation volume factor at the pressure p_O is calculated using the formula:

$$B_O(p_O) = \frac{B_O(p_{\text{ref}})\mu_O(p_{\text{ref}})exp(-(c_O - C_v)(p_O - p_{\text{ref}}))}{\mu_O(p_O)}.$$

Default: none

```
Example
TABDIMS
1 2 2* 1
/
...
PVDCO
  83.20  1.15  1.7e-4 0.22 0
 /
 102.34  1.15  1.56e-5 0.73 0
 267.67  1.11  2.89e-5 0.69 0
 /
```

In this example there are two regions with different dead oil PVT properties. Note that the keyword TABDIMS (see 6.1.13) sets the number of such regions equal to 2.

6.5.3 PVTO

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets live oil PVT properties for all PVT regions. The following parameters (one set of data) are to be specified:

- gas-oil ratio (this is $R_{G,O}$ in 2.13),

- bubble point pressure (this is argument p_O in 2.9, 2.13, 2.8),

- oil formation volume factor (this is B_O in 2.9),

- oil viscosity at bubble point pressure (this is μ_O in 2.8).

Each set of data should be terminated by a slash /
For each PVT region oil PVT properties tables should follow, each table must be terminated by a slash /. There must be the same number of tables as number of PVT regions defined in TABDIMS (see 6.1.13).
A number of sets of data in one PVT table (number of different $R_{G,O}$) shouldn't be larger than 6-th parameter of TABDIMS (see 6.1.13).

One set of data specifies PVT properties for one gas-oil ratio $R_{G,O}$. Some sets of data may contain extra data specifying properties of undersaturated oil for given gas-oil ratio. This extra data must be entered in each table for the largest $R_{G,O}$. The number of lines of this data for one $R_{G,O}$ shouldn't be larger than 4-th parameter of TABDIMS (see 6.1.13)).
For undersaturated part of the table, solution gas-oil ratio $R_{G,O}$ is not specified. Thus the extra data should be entered as 3 columns (contain 2-nd, 3rd and 4th parameter described above).
Default: none

```
Example
TABDIMS
1 3 2* 1
/
...
PVTO
 1         5   1.031  5.81  /
 12.33    52   1.080  5.03  /
 21.65    73  1.1021  4.23
          204   1.092  4.62
          321   1.016  6.02  /
 /
 1         5  1.0002  3.58  /
 14.87    58   1.086  2.93  /
 27.7     90   1.113  2.25
          234     1.1  2.88
          387   1.121  3.96  /
 /
 1         5  1.0002  3.58  /
 18.67    57  1.0730  2.89  /
 31.65    88  1.1083   2.2
          248   1.093  2.57
          334   1.073  4.23  /
 /
```

This example sets different live oil PVT properties for three PVT regions: the first table contains PVT properties of the first PVT region, the second table — of the second one, the third table — of the third one. Note that TABDIMS (see 6.1.13) keyword defined 3 PVT regions. The initial bubble point pressure $P_{bub\ user}$ is 73 for the first PVT region, 90 for the second and 88 for the third.

6.5.4 PVTW

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets water PVT properties for all PVT regions. The following parameters are to be specified:

- reference pressure, (this is argument p_{ref} in 2.9, 2.8),

- water formation volume factor, (this is B_W in 2.9),

- water compressibility, (this is c_W in 2.9),

- water viscosity (this is μ_W in 2.8),

- water viscosibility $C_v = \dfrac{\partial \mu_W}{\partial P} \mu_W$.

For each PVT region water PVT properties should follow, each line must be terminated by a slash /. There must be the same number of strings as number of PVT regions defined in TABDIMS (see 6.1.13).

Water formation volume factor for given pressure p_W will be calculated as follows

$$B_W(p_W) = \frac{B_W(p_{\text{ref}})}{1 + c_W(p_W - p_{\text{ref}}) + c_W^2(p_W - p_{\text{ref}})^2/2}.$$

Default: none

```
Example
TABDIMS
1 2 2* 1
/
...
PVTW
234.00  1.02  4.0E-0005  0.42
/
250.00  1.00  4.0E-0005  0.43
/
```

This example considers two PVT regions. First line after **PVTW** sets water PVT properties for the first region, the second line — for the second one.

6.5.5 PVCO

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets live oil (in compressibility form) PVT properties. These properties can also be specified by the keyword PVTO (see 6.5.3). The difference between PVCO and PVTO (see 6.5.3) is following: PVCO is used if undersaturated oil with a special $R_{G,O}$ (Ч 2.13) has a pressure-independent compressibility and the derivative of viscosity of undersaturated oil is pressure-independent too. Hence, tables for undersaturated oil which set the dependence between the formation volume factor, viscosity and pressure shouldn't be specified.

Live oil PVT properties for each region should be entered in a new line. Region properties should be terminated with a slash /. The number of lines is equal to the number of regions with different PVT properties (set by the second parameter of the keyword TABDIMS (see 6.1.13)).

The following parameters are to be specified:

- the bubble point pressure for oil with gas-oil ratio specified by the 2-nd parameter of PVCO (p_O in 2.9, 2.13, 2.8),

- gas-oil ratio of saturated oil with buble point pressure specified by the 1-st parameter of PVCO ($R_{G,O}$ in 2.13),

- the formation volume factor of saturated oil at the bubble point pressure (B_O in 2.9),

- the viscosity of saturated oil at the bubble point pressure (μ_O in 2.8),

- the compressibility of undersaturated oil with gas-oil ratio specified by the 2-nd parameter of PVCO ($C = -\dfrac{\partial B_O}{\partial P} \dfrac{1}{B_O}$).

- the viscosibility of undersaturated oil with gas-oil ratio specified by the 2-nd parameter of PVCO ($C_v = \dfrac{\partial \mu_O}{\partial P} \dfrac{1}{\mu_O}$).

```
Example
TABDIMS
1 2 2* 1
/
...
PVCO
 800   0.15  1.034  1.16  1.2e-5   1
 1000  0.35  1.039  1.13  1.7e-5   1
 /
 600   0.12  1.012  1.14  1.05e-5  1
 1200  0.34  1.15   1.18  1.8e-5   1
 /
```

This example sets live oil PVT properties for two PVT regions. Note that TABDIMS (see 6.1.13) keyword defined 2 PVT regions.

6.5.6 PVDG

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets gas PVT properties for all PVT regions. The following parameters are to be specified:

- reference pressure, (this is argument p_G in 2.9, 2.8),

- gas formation volume factor, (this is B_G in 2.9),

- gas viscosity (this is μ_G in 2.8).

For each PVT region gas PVT properties tables should follow, each table must be terminated by a slash /. There must be the same number of tables as number of PVT regions defined in TABDIMS (see 6.1.13).

Default: none

```
Example
TABDIMS
1 3 2* 1
/
...
PVDG
 5    0.117    0.0118
 73   0.0109   0.0142
 204  0.00408  0.021
 /
 5    0.117    0.0117
 90   0.0109   0.0149
 234  0.00408  0.0251
 /
 5    0.117    0.0117
 88   0.0109   0.0147
 248  0.00408  0.0268
 /
```

This example considers three PVT regions with different gas PVT properties: the first table sets gas PVT properties for the first region, the second one — for the second one, and the third — for the third one.

6.5.7 PVTG

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets wet gas PVT properties for all PVT regions. The following parameters (one set of data) are to be specified:

- dew point pressure (this is argument p_G in 2.9, 2.14, 2.8),

- oil vaporization in gas (this is $R_{O,G}$ in 2.14),

- gas formation volume factor (this is B_G in 2.9),

- gas viscosity at bubble point pressure (this is μ_G in 2.8).

Each set of data should be terminated by a slash /
For each PVT region oil PVT properties tables should follow, each table must be terminated by a slash /. There must be the same number of tables as number of PVT regions defined in TABDIMS (see 6.1.13).
A number of sets of data in one PVT table (number of different p_G) shouldn't be larger than 7-th parameter of TABDIMS (see 6.1.13).

One set of data specifies PVT properties for one dew point pressure p_G. Some sets of data may contain extra data specifying properties of undersaturated gas for given dew point pressure. This extra data must be entered in each table for the largest p_G. The number of lines of this data for one p_G shouldn't be larger than 4-th parameter of TABDIMS (see 6.1.13)).
For undersaturated part of the table, dew point pressure p_G is not specified. Thus the extra data should be entered as 3 columns (contain 2-nd, 3rd and 4th parameter described above).

Initial oil vaporization in gas is to be specified by keyword RV (see 6.9.20). For pressures above dew point pressure saturated oil properties are extrapolated linearly.

Default: none

```
Example
TABDIMS
1 2 2* 1
/
...
PVTG
 1        0.00007   1.131  0.0120  /
 12.33    0.00006   0.080  0.0123  /
 21.65    0.00005  0.1021  0.0126  /
 250.0    0.00006  0.0092   0.021
              0.   0.0098    0.02  /
 /
 1        0.000017  1.131  0.0120  /
 12.33    0.000016   0.080  0.0123  /
 21.65    0.000015  0.1021  0.0126  /
 230.0   0.0000155  0.0092   0.021
              0.   0.0098    0.02  /
 /
```

This example sets different live oil PVT properties for two PVT regions: the first table contains PVT properties of the first PVT region, the second table — of the second one. Note that TABDIMS (see 6.1.13) keyword defined 2 PVT regions.

6.5.8   PVZG

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets dry gas PVT properties (compressibility factor is taken into consideration). The data should consist of tables (number of tables less or equal to the 2-nd parameter of the keyword TABDIMS (see 6.1.13).) Each table contains two sets of data. A set of data must be terminated by a slash /.

1-st set of data:

One parameter is to be specified:

- the reference temperature (for this table). This temperature is used in the formula of conversion compressibility factor to formation volume factor.

2-nd set of data (a table consists of following columns):

Following parameters (one row of the table) are to be specified:

- gas phase pressure,

- compressibility factor (Z-factor) for the pressure given by 1-st parameter,

- gas viscosity for the pressure given by 1-st parameter (μ_G in 2.8).

The number of rows in this table should be equal or greater than 2 and not greater than the 4-th parameter of the keyword TABDIMS (see 6.1.13).

This formula represent a connection among the formation volume factor B_G (B_G in 2.9), the reference temperature T_{ref} and the pressure p:

$$B_G = Z * \left(\frac{T_{ref} + T_{base}}{T_s + T_{base}} \right) * \frac{p_s}{p}$$

ЗДЕ Z - compressibility factor (Z - factor), $p_s = 1.103$ Barsa, $T_s = 15.56°\,\mathrm{K}$ - pressure and temperature at standard conditions. $T_{base} = 273.105°\,\mathrm{K}$ -absolute temperature.

Default: nothing

Example
```
TABDIMS
1 2 2* 1
/
...
PVZG
 120       /
 1000  1.31  0.0120  /
 1200  1.39  0.0140  /
 1600  1.48  0.0160  /
 2000  1.56  0.0180  /
 /
```

In this example the reference temperature is equal to 120. The 2-nd item of data sets gas phase pressure, compressibility factor and gas viscosity.

6.5.9   RSCONST

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets a constant and uniform gas concentration in dead oil. If there is no gas phase and a pressure is always above the bubble point a black oil modeling can be more efficient.

The keywords GAS (see 6.1.28) and DISGAS (see 6.1.31) shouldn't be specified in the Definition section (6.1). So gas is not considered as an active phase. The model becomes two-phase or single-phase (if water phase is presented or not).

The data should be terminated by a slash /. Following parameters are to be specified:

1. dissolved gas concentration (constant and uniform);

2. bubble point pressure (the run will stop if the pressure in any block falls below this value);

The specification of this keyword leads to the modification of the oil density (there is the dissolved gas in oil phase).

So gas flow rate is equal to the product of the oil flow rate and dissolved gas concentration.

The keyword RSCONSTT (see 6.5.10) sets a constant and uniform gas concentration in dead oil for all PVT regions (this concectration may be different for different PVT regions).

```
Example
RSCONST
0.24 980
/
```

This example sets dissolved gas concentration to 0.24 and bubble point pressure — 980.

6.5.10   RSCONSTT

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword sets a constant and uniform gas concentration in dead oil for every PVT region. PVT regions are specified using the keyword PVTNUM (see 6.4.1). If there is no gas phase and a pressure is always above the bubble point a black oil modeling can be more efficient.

The keywords GAS (see 6.1.28) and DISGAS (see 6.1.31) shouldn't be specified in the Definition section (6.1). So gas is not considered as an active phase. The model becomes two-phase or single-phase (if water phase is presented or not).

The difference between RSCONST (see 6.5.9) and RSCONSTT is the following: RSCONSTT is used to set a constant and uniform gas concentration in dead oil for all PVT regions (this conccetration may be different for different PVT regions). Oils with different gas concentrations shouldn't mix (transmissibility shouldn't be allowed between such PVT regions). Otherwise the gas concentration will change and the model should be considered as a model with active gas phase.

The keyword is followed by data lines. Each line should be terminated by a slash /. The number of lines is equal to the number of PVT regions (second parameter of the keyword TABDIMS (see 6.1.13)). Following parameters are to be specified for every line:

1. dissolved gas concentration (constant and uniform);

2. bubble point pressure (the run will stop if the pressure in any block falls below this value);

The specification of this keyword leads to the modification of the oil density (there is the dissolved gas in oil phase).

So gas flow rate is equal to the product of the oil flow rate and dissolved gas concentration (of the PVT region to which this well belongs to). The gas flow rate of a group of wells (or reservoir) is calculated as the sum of wells' gas flow rates of this group (or reservoir).

The second parameter of TABDIMS (see 6.1.13) is equal to 3.

```
Example
RSCONSTT
 0.24   980  /
 0.31  1017  /
 0.42  1025  /
```

This example sets dissolved gas concentration to 0.24 and bubble point pressure — 980 (first PVT region), 0.31 and 1017 (second PVT region), 0.42 and 1025 (third PVT region).

6.5.11 RVCONST

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword sets a constant and uniform oil concentration in dry gas. If there is no oil phase and a pressure is always above the dew point a modeling of gas systems can be more efficient.

The keywords OIL (see 6.1.27) and VAPOIL (see 6.1.30) shouldn't be specified in the Definition section (6.1). So oil is not considered as an active phase. The model becomes two-phase or single-phase (if water phase is presented or not).

The data should be terminated by a slash /. Following parameters are to be specified:

1. vaporized oil concentration (constant and uniform);

2. dew point pressure (the run will stop if the pressure in any block falls below this value);

The specification of this keyword leads to the modificaton of the gas density (there is the vaporized oil in gas phase).

So oil flow rate is equal to the product of the gas flow rate and vaporized oil concentration.

```
Example
RVCONST
0.00024 380
/
```

This example sets vaporized oil concentration to 0.00024 and dew point pressure — 380.

6.5.12 ROCK

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets rock compressibility for all PVT regions. The following parameters are to be specified:

- reference pressure,

- rock compressibility coefficient (this is coefficient c in 2.5).

Number of strings depends on number of PVT regions defined in TABDIMS (see 6.1.13).
Default: none

```
Example
TABDIMS
1 2 2* 1
/
...
ROCK
234.00  4.0E-0005
/
250.00  4.0E-0005
/
```

In this examples two PVT regions are considered (TABDIMS (see 6.1.13)). The first line after ROCK keyword sets compressibility for the first PVT region, the second line — for the second one.

6.5.13 ROCKOPTS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to set options for rock compaction or rock compressibility. The data should be terminated with a slash /.

The following parameters should be specified:

1. The method of application of the keyword OVERBURD (see 6.5.14) to the pressures (specified via ROCKTAB (see 6.2.10)). Two options are possible: PRESSURE or STRESS.
   STRESS option. If the overburden pressure ($P_{overburden}$ − OVERBURD (see 6.5.14)) is greater than the fluid pressure (P_{fluid}) (which is often true), the effective fluid pressure will be negative ($P_{effluid} = P_{fluid} - P_{overburden}$, $P_{effluid}$ - effective fluid pressure). In this case the rock compaction tables (ROCKTAB (see 6.2.10)) will have to be entered with negative pressure values. One can avoid this using STRESS option. Pressure values in this case can be entered positive: pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, which will be positive ($P_{efover} = P_{overburden} - P_{fluid}$).
   If the keyword OVERBURD (see 6.5.14) isn't used, one should left defaulted this parameter.

2. reference pressure option STORE or NOSTORE.
   STORE: copies the initial equilibrated pressure at the start of the run into the overburden array. This has the effect of referencing the pore volume (as the function of pressure) to the initial pressure rather than the reference pressure, that is, the pressure with a pore volume multiplier of 1.0 in ROCKTAB (see 6.2.10) tables. Thus the input pore volume for a cell is defined as the pore volume at initial conditions rather than the pore volume at the reference pressure. OVERBURD (see 6.5.14) should not be used with the STORE option as the input OVERBURD (see 6.5.14) data will be overwritten and ignored.
   If STORE option is used with the keyword ROCK (see 6.5.12) for the elastic rock compressibility model, the input pore volume for cell is similarly defined as the reservoir pore volume at initial conditions rather than the pore volume at the reference pressure.

3. table region to be used: SATNUM, PVTNUM, ROCKNUM.

4. this is an Eclipse compatibility field.

Default:

1. PRESSURE,

2. NOSTORE,

3. PVTNUM.

Example
```
ROCKOPTS STRESS /
```

6.5.14 OVERBURD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies rock overburden pressure tables. The keyword can be used if the number of rock regions is specified (ROCKCOMP (see 6.1.19)).

The data consists of tables of rock overburden data (the number of tables should be equil to the 2-nd parameter of the keyword ROCKCOMP (see 6.1.19)). Each table should be terminated with a slash /. Rock regions are specified via ROCKNUM (see 6.4.6).

Each table row should contain the following parameters:

1. the depth,

2. the corresponding overburden pressure value.

Default: if the keyword isn't present, the overburden pressure is taken to be zero.

STRESS option (the 1-st parameter of the keyword ROCKOPTS (see 6.5.13)). If the overburden pressure ($P_{overburden}$ – OVERBURD (see 6.5.14)) is greater than the fluid pressure (P_{fluid}) (which is often true), the effective fluid pressure will be negative ($P_{effluid} = P_{fluid} - P_{overburden}$, $P_{effluid}$ - effective fluid pressure). In this case the rock compaction tables (ROCKTAB (see 6.2.10)) will have to be entered with negative pressure values. One can avoid this using STRESS option. Pressure values in this case can be entered positive: pore volume and transmissibility multipliers will be tabulated against the effective overburden pressure, which will be positive ($P_{efover} = P_{overburden} - P_{fluid}$).

```
Example
OVERBURD 1800 570
5000 1600
7000 2100
9000 2650
/
```

In this example overburden pressure if specified for one rock region.

6.5.15 DENSITY

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets fluid densities for all PVT regions. The following three parameters are to be specified:

- oil density (this is $\rho_{O,SC}$ in 2.11),

- water density (this is $\rho_{W,SC}$ in 2.11),

- gas density (this is $\rho_{G,SC}$ in 2.11).

Number of strings depends on number of PVT regions defined in TABDIMS (see 6.1.13).

Default: 600 kg/m^3 999.014 kg/m^3 1 kg/m^3

```
Example
TABDIMS
1 2 2* 1
/
...
DENSITY
600 999.014 1.0
/
883.40 1001.00 0.90
/
```

In this example there are two PVT regions with different properties: the first line sets fluid densities for the first region, the second line — for the second one.

6.5.16 NPROPANTS

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keywords sets the number of proppants in the model. The data should be terminated with a slash `/`.

Example
NPROPANTS
2/

In this example there are 2 proppants.

6.5.17   PROPANTNAMES

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets proppant names for all proppants in the model. The same number of names should be specified as the number of proppants in the model (NPROPANTS (see 6.5.16)). The data should be terminated with a slash /.

```
Example
NPROPANTS
2/
PROPANTNAMES
'propant 12/18' 'propant 16/20' /
```

In this example 2 proppants are specified. 1-st proppant name is 'propant 12/18', 2-nd proppant name – 'propant 16/20'.

6.5.18   PROPANTTABLE

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets the table of relation between pressure ant proppant permeability.

The number of columns of this table depends on the number of proppants (keyword NPROPANTS (see 6.5.16)).

One table row consists of the following parameters:

1. pressure (FIELD: psi, METRIC: bar);

2. proppant permeability (proppant 1) for the unit of hydraulic fraction lenght at this pressure;

3. proppant permeability (proppant 2) for the unit of hydraulic fraction lenght at this pressure;

4. ...

5. proppant permeability (proppant NPROPANTS) for the unit of hydraulic fraction lenght at this pressure.

An arbitrary number of table rows could be specified with different pressures. The data should be terminated with a slash /.

Default:

Pressure values couldn't be defaulted.
Permeability values couldn't be defaulted in the first table row. It should be at list 2 non-defaulted values of permeability. If the value is defaulted the result of linear interpolation will be used in calculations. Permeability value for the last pressure value in the table should be zero for all propants (pressure of fraction closing).

```
Example
NPROPANTS
2/
PROPANTNAMES
'propant 12/18' 'propant 16/20' /
PROPANTTABLE
100 10600 5100
200 * 3900
400 2400 1800
800 200 0
1000 0 *
/
```

In this example 2 proppants are specified. 1-st proppant name is 'propant 12/18', 2-nd proppant name – 'propant 16/20'.

The table of proppant properties is specified for these two proppants (5 different pressure values in the table). Default: permeability of 1-st proppant at pressure 200 and permeability of 2-nd proppant at pressure 1000.

6.5.19   FLOWFUNC

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

   The keyword specifies the coefficients of flow functions (the dependence between permeability and flow (or time) in the model 5.4.8).

   Function f of dimensionless flow s (a ration of accumulated flow from block to block pore volume), is the following

1. $f(0) = 1$;

2. $f(s) \geq 0$ for all s.

There are 3 flow function types:

1. Exponential: $f(s) = k + (1-k) * \exp(-a * s)$, Parameters: $k \geq 0$, $a > 0$ are specified via this keyword FLOWFUNC (see 6.5.19) (function type EXP).

2. Linear: $f(s) = \max\{1 + (k-1) * a * s, 0\}$, Parameters: $k \geq 0$, $a > 0$ are specified via this keyword FLOWFUNC (see 6.5.19) (function type LIN).

3. Tabular: the table $(s, f(s))$ is specified via the keyword FLOWFTAB (see 6.5.22) (the number of tabular functions are specified via the keyword NFLOWFTB (see 6.5.20), tabular function names – FLOWFNAMES (see 6.5.21)).

If function is not specified, then $f(s) = 1$ is considered.

   One table row contains the parameters for one function:

1. function name;

2. function type (LIN - linear, EXP - exponential);

3. coefficient k;

4. coefficient a.

   Each row should be terminated with a slash /. The number of rows should be specified the same as the sum of linear and exponential functions in the model. The data should be terminated with a slash /.

```
Example
FLOWFUNC
'Func 1' LIN 8 18.5 /
'Func 2' LIN 1.2 4.7 /
'Func 3' EXP 4 2 /
'Func 4' EXP 3.6 3 /
/
```

In this example there are coefficients for 4 flow functions ('Func 1' and 'Func 2' - linear, 'Func 3' and 'Func 4' - exponential).

6.5.20 NFLOWFTB

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the number of tabular flow functions (the dependence between permeability and flow (or time) in the model 5.4.8). The data should be terminated with a slash /.

Function $f(s)$ is the following:

1. $f(0) = 1$;

2. $f(s) \geq 0$ for all s.

Example
NFLOWFTB
2/

In tis example there are 2 tabular flow functions.

6.5.21 FLOWFNAMES

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the names of tabular flow functions (the dependence between permeability and flow (or time) in the model 5.4.8). The same number of names should be specified as the number of flow functions in the model (NFLOWFTB (see 6.5.20)). The data should be terminated with a slash /.

```
Example
NFLOWFTB
2/
FLOWFNAMES
'Func 5' 'Func 6' /
```

In this example there are 2 tabular flow functions: 'Func 5' and 'Func 6'.

6.5.22   FLOWFTAB

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the table of flow functions (the dependence between permeability and flow (or time) in the model 5.4.8).

Function $f(s)$ is the following:

1. $f(0) = 1$;

2. $f(s) \geq 0$ for all s.

The number of table columns depends on the number of flow functions in the model (the keyword NFLOWFTB (see 6.5.20)).

One table row consists of the following parameters:

1. argument s value;

2. function (number 1) value at the given argument s value;

3. function (number 2) value at the given argument s value;

4. ...

5. function (number NFLOWFTB) value at the given argument s value.

An arbitrary number of table rows could be specified with different argument s values. The data should be terminated with a slash /.

Default:
Argument s values couldn't be defaulted.
Function values couldn't be defaulted in the first table row. It should be at list 2 non-defaulted values.
If the value is defaulted the result of linear interpolation will be used in calculations. The first value of argument s should be 0. The first function value (for all functions) at zero argument s value should be 1.

Example
```
NFLOWFTB
2/
FLOWFNAMES
'Func 5' 'Func 6' /
FLOWFTAB
0 1 1
1 0.5 *
2 * 0.5
3 0.1 0.1
/
```

In this example there are 2 tabular flow functions: 'Func 5' and 'Func 6'. The table consists of 4 rows (4 different argument s values).

6.5.23 MAPEXPER

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to provide calculations with minimal data on disc. Parameters of this keyword can be each name of maps in the section where the keyword is. Allowed sections: GRID, EDIT, PROPS, SOLUTION.

Any number of data records can be specified (one data record corresponds to one modified map). Each data record should be terminated with a slash /. All data record should be terminated with a final slash /.

One data record consists of the following parameters:

1. 'Map name' - name of modified map. Allowed names depend on section where the keyword is. Allowed sections: GRID, EDIT, PROPS, SOLUTION.

2. 'Operation' - name of applied operation. Each operation is binary and has arguments:

   - first operand – the map which is specified by parameter 1 of this keyword,
   - second operand – pseudorandom number in the range between parameters 3 and 4; this number is specified via the rule determined by parameters 5-10.

   The result of operation is placed to the map specified by parameter 1. Allowed operations:

   - 'MULT' - multiplication
   - 'DIV' - division
   - 'ADD' - addition
   - 'SUB' - subtraction

   Default: 'MULT'.

3. 'min var' - real number, which sets the minimal threshold of map modification (map is specified via parameter 1.)
   Default: 1 (there is no decreasing modification).

4. 'max var' - real number, which sets the maximal threshold of map modification (map is specified via parameter 1.)
   Default: 1 (there is no increasing modification).

5. 'seed' - integer, which specifies first pseudorandom number. It can be identifier of variant.

6. '1D rand type' - distribution type of one-dimensional chance quantity. Possible variants:

- 'U' - uniform (uniform distribution)
- 'G' - Gauss (normal distribution)

Default: 'U'

7. '3D rand type' - distribution type of three-dimensional chance quantity. Possible variants:

- 'C' - constant (piecewise constant distribution: the same value of pseudorandom number in each block (belongs to one region) is used (regions are specified via parameter 10); this region number should be between numbers of regions specified by parameters 8 and 9; for blocks from other regions an array of parameter 1 doesn't change)
- 'R' - random (random distribution: in each block it's own pseudorandom number is used (regions are specified via parameter 10); region number should be between numbers of regions specified by parameters 8 and 9; for blocks from other regions an array of parameter 1 doesn't change)

Default: 'C'

8. 'min reg' - minimal value of region number in the array specified by parameter 10. **Default:** no limit.

9. 'max reg' - maximal value of region number in the array specified by parameter 10. **Default:** no limit.

10. 'Region name' - array which specifies region number. Possible variants:

- 'M' - MULTNUM
- 'F' - FLUXNUM
- 'O' - OPERNUM

Default: 'M'

Example
```
MAPEXPER
'NTG' 'MULT' 0.1 1* 123456 /
'MULTPV' 'DIV' 0.1 1* 123456 /
/
```

In this example in each grid block (since there is no limits of minimal and maximal region number are not specified) is specified an operation: multiplication of NTG by the number in range [0.1, 1] and division of MULTV by the same number (since the values of variants are the same). So reserves in these experiments are equal. Arrays NTG (see 6.2.18), MULTPV (see 6.2.21) have defauls values and so they cound be not specified before the keyword MAPEXPER (see 6.5.23).

Example
```
MAPEXPER
'PERMX' 'MULT' 0.1 10 123456 2* 3 5 'O' /
'PERMY' 'MULT' 0.1 10 123456 2* 3 5 'O' /
'PERMZ' 'MULT' 0.1 10 123456 2* 3 5 'O' /
/
```

In this example in each grid block with OPERNUM (see 6.4.11) region number in the range between 3 and 5 there is an operation of multiplication of X, Y, Z permeabilities by the same number.

6.5.24 TRACER

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar	
✓ eclipse 100	☐ CMG imex	☐ CMG stars	

The keyword sets passive tracers. In e100 each tracer is associated with one of three phases (water, oil, gas). In e300 each tracer is associated with one hydrocarbon component or water. In tNavigator each tracer is associated with one hydrocarbon component or water.

For each tracer the data line should be specified. Each line should be terminated with a slash /. The data should be terminated with a final slash /.

Each line should consist of the following parameters:

1. tracer name (in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters),

2. the name of the fluid associated with the tracer: e100 – OIL, WAT, GAS; e300 – hydrocarbon component name or WATER,

3. units for tracer amount (this parameter is optional and is used for reporting only),

4. ignored, this is an Eclipse compatibility field(this parameter is specified for partitioned tracers only),

5. ignored, this is an Eclipse compatibility field(this parameter is specified for partitioned tracers only).

```
Example
TRACER
A WAT /
B WAT /
C OIL /
/
```

In this example there are 3 tracers: tracers A and B are associated with water, tracer C – with oil.

6.5.25 TRACERM

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets passive tracers for which the holding time in the reservoir will be calculated.

Tracer name should be specigied in its own new line and be terminated with a slash /. The data should be terminated with a final slash /.

```
Example
TRACER
A WAT /
B WAT /
/

...
TRACERM
A /
B /
/
```

In this example there are 2 tracers A and B for which the holding time in the reservoir will be calculated.

6.5.26 SALTPROP

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets properties of dissolved and reservoir salt. The keyword can be used only for runs in which the Brine option is active (the keyword BRINE (see 6.1.33)).

Number of data records depends on number of PVT regions (defined in TABDIMS (see 6.1.13)). Each data record contains data for one PVT region and should be terminated with a slash /. All data should be terminated with a final slash /.

One data record consists of the following parameters:

1. ultimate concentration of dissolved (in the water) salt (concentration of saturated salt solution);

2. density of reservoir salt;

3. solution rate constant of reservoir salt ϑ_{salt}.

Then the solution rate of reservoir salt v_{salt} (kg/day) in the volume V will be calculated via formula:

$$v_{salt} = \vartheta_{salt} \left(m_{max} - m_{cur} \right),$$

where m_{cur} – current salt quantity in the solution (kg), m_{max} – salt quantity in the saturated salt solution (kg) in the volume V.

Default: 300 kg/m^3 2165 kg/m^3 0 (1/day).

Example
```
SALTPROP
350 2165 0.0001
/
```

6.5.27 SALTTRM

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the dependence between permeability and amount of dissolved reservoir salt. The keyword can be used only for runs in which the Brine option is active (the keyword BRINE (see 6.1.33)).

Number of tables depends on number of PVT regions (defined in TABDIMS (see 6.1.13)). Each table contains data for one PVT region and should be terminated with a slash /. All data should be terminated with a final slash /.

One table row consists of the following parameters:

1. ratio of the volume of dissolved reservoir salt to the initial volume of rock in the cell;

2. absolute permeability multiplier.

Example
```
SALTTRM
0.0 1
0.01 10
0.1 100
0.4 400
0.5 500
/
```

In this example one table is specified for one PVT region.

6.5.28 TRMMULTC

	tNavigator		eclipse 300		MORE Roxar
✓					
	eclipse 100		CMG imex		CMG stars

The keyword sets the dependence between absolute permeability multiplier and tracer concentration (k_{conc} in 2.17), for which the option of calculation the holding time in the reservoir in enable, (keyword TRACERM (see 6.5.25)).

One should specify tracer name and the following data:

1. tracer concentration (T_{conc});

2. function $k_{conc}(T_{conc})$.

Then the final permeability multiplier will be calculated via the formula:

$$k_{mult} = 1 - (1 - k_{conc}(T_{conc}))(1 - k_{time}(t)),$$

where $k_{time}(t)$ is set using the keyword TRMMULTT (see 6.5.29). Dependences between absolute permeability multiplier and tracer concentration should be entered as a table and be terminated with a slash /. The data should be terminated with a final slash /.

```
Example
TRACERM
A /
B /
/

...
TRMMULTC
A
0.1 1.0
0.2 0.9
0.5 0.3 /
B
0.1 0.5
0.3 0.1 /
/
```

In this example the dependence between absolute permeability multiplier and tracer concentration is specified for two tracers A and B.

6.5.29 TRMMULTT

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the dependence between absolute permeability multiplier and holding time in the reservoir (keyword TRACERM (see 6.5.25)).

One should specify tracer name and the following data:

1. holding time in the reservoir (t);

2. function $k_{time}(t)$.

Then the final permeability multiplier will be calculated via the formula:

$$k_{mult} = 1 - (1 - k_{conc}(T_{conc}))(1 - k_{time}(t)),$$

where $k_{conc}(T_{conc})$ is set using the keyword TRMMULTC (see 6.5.28). Dependences between absolute permeability multiplier and holding time should be entered as a table and be terminated with a slash /. The data should be terminated with a final slash /.

```
Example
TRACERM
A /
B /
/

...
TRMMULTT
A
0 1.0
1000 0.9 /
/
B
0 1.0
1500 0.8 /
/
```

In this example the dependence between absolute permeability multiplier and holding time in the reservoir is specified for two tracers A and B.

6.5.30   PVTWSALT

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword supplies the water PVT data for runs in which the Brine option is active (the keyword BRINE (see 6.1.33)). The keyword is used in place of PVTW (see 6.5.4).

The data consists of tables. The number of tables is equal to the second parameter of the keyword TABDIMS (see 6.1.13). Each table consists of two data records, each terminated by a slash (/).

First data record:

1. reference pressure (P_{ref}) for this table (*barsa*);

2. reference salt consentration for stock tank water (kg/m^3).

Second data record (one table row consists of):

1. the salt consentration (kg/m^3);

2. the water formation volume factor at the reference pressure as a function of salt consentration $B_w(P_{ref})$ (rm^3/sm^3);

3. the water compressibility $C = -(\frac{\partial B_w}{\partial P})/B_w$ ($1/bars$);

4. the water viscosity at the reference pressure $\mu_w(P_{ref})$;

5. the water viscosibility $C_v = (\frac{\partial \mu_w}{\partial P})/\mu_w$ ($1/bars$).

```
Example
PVTWSALT
250 0.0 /
0.00 1.00 4E-005 0.52 0.0
160.00 0.98 8E-005 0.52 0.0
/
```

6.6 Relative permeabilities and capillary pressures

This section describes keywords used to define relative permeabilities and capillary pressures. First tables used to calculate phase relative permeabilities in case of multiphase flow, and to calculate phase capillary pressures are described. Next keywords regulating relative permeabilities interpolation methods and end-point scaling are listed.

The keywords specifying relative permeabilities (see Relative permeabilities and capillary pressures (6.6)):

- SWOF (see 6.6.1) — oil-in-water relative permeability (both oil and water phases should be present);

- SGOF (see 6.6.2) — oil-in-gas relative permeability (both oil and gas phases should be present);

- SWFN (see 6.6.4) — water saturation function;

- SGFN (see 6.6.5) — gas saturation function;

- SOF2 (see 6.6.3) — oil saturation function (only for two-phase models);

- SOF3 (see 6.6.6) — oil saturation function (only for three-phase models);

- SGWFN (see 6.6.7) — gas-water saturation function (may be used in gas-water models, when oil isn't present).

The are two keyword groups; one shouldn't mix keywords from different groups.

group 1: SWOF (see 6.6.1), SGOF (see 6.6.2)

group 2: SWFN (see 6.6.4), SGFN (see 6.6.5), SOF2 (see 6.6.3), SGWFN (see 6.6.7), SOF3 (see 6.6.6)

One can see different keyword combinations in Property section (6.5) (PVT properties, permeabilities, initial pressures and saturations).

6.6.1 SWOF

	tNavigator		eclipse 300		MORE Roxar
✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
✓	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies relative permeability tables for all saturation function regions from TABDIMS (see 6.1.13) for water-oil systems. Each table is a 4-column table with the following columns:

1. SW (water saturation) (this is argument value for functions below)

2. KRWO (water permeability) (this is function k_{rWO} in 2.6)

3. KROW (oil permeability) (this is function k_{rOW} in 2.6)

4. POW (oil-water capillary pressure) (this is function P_{cOW} in 2.12.2)

Number of tables depends on number of saturation function regions defined in TABDIMS. Each table must be terminated by a slash /.
   Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SWOF
 0      0      1      0
 0.25   0      1      0
 0.3    0.002  0.81   0
 0.35   0.008  0.64   0
 0.4    0.018  0.49   0
 0.45   0.032  0.36   0
 0.5    0.05   0.25   0
 0.55   0.072  0.16   0
 0.6    0.098  0.09   0
 0.65   0.128  0.04   0
 0.7    0.162  0.01   0
 0.75   0.2    0      0
 1      0.2    0      0
 /
```

This example sets relative permeability for a two-phase water-oil system for one saturation function region.

Example
```
TABDIMS
2 1 2* 1
/
...
SWOF
 0.42      0           0.737      0
 0.45364  1.1e-005   0.704875   0
 0.48728  0.000225   0.610213   0
 0.52092  0.001292   0.468702   0
 0.55456  0.00438    0.310527   0
 0.5882   0.011044   0.170133   0
 0.62184  0.023012   0.072027   0
 0.65548  0.04193    0.020856   0
 0.68912  0.069122   0.003178   0
 0.72276  0.10541    0.000115   0
 0.7564   0.151      0          0
 0.79004  0.205444   0          0
 0.82368  0.267672   0          0
 0.85732  0.336083   0          0
 0.89096  0.408671   0          0
 0.9246   0.483175   0          0
 0.95824  0.557237   0          0
 1        0.645099   0          0
 /
 0      0      1      0
 0.25   0      1      0
 0.3    0.002  0.81   0
 0.35   0.008  0.64   0
 0.4    0.018  0.49   0
 0.45   0.032  0.36   0
 0.5    0.05   0.25   0
 0.55   0.072  0.16   0
 0.6    0.098  0.09   0
 0.65   0.128  0.04   0
 0.7    0.162  0.01   0
 0.75   0.2    0      0
 1      0.2    0      0
 /
```

This example sets relative permeabilities for a two-phase water-oil system for two saturation function regions.

6.6.2 SGOF

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies relative permeability tables for two-phase gas-oil systems for all saturation regions from TABDIMS (see 6.1.13). Each table is a 4-column table with the following columns:

1. SG (gas saturation) (this is argument value for functions below)

2. KRGO (gas permeability) (this is function k_{rGO} in 2.6)

3. KROG (oil permeability) (this is function k_{rOG} in 2.6)

4. POG (oil-gas capillary pressure) (this is function P_{cOG} in 2.12.1)

Number of tables depends on number of saturation regions defined in TABDIMS. Each table must be terminated by a slash /.

Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SGOF
 0      0      1      0
 0.25   0      1      0
 0.3    0.002  0.81   0
 0.35   0.008  0.64   0
 0.4    0.018  0.49   0
 0.45   0.032  0.36   0
 0.5    0.05   0.25   0
 0.55   0.072  0.16   0
 0.6    0.098  0.09   0
 0.65   0.128  0.04   0
 0.7    0.162  0.01   0
 0.75   0.2    0      0
 1      0.2    0      0
 /
```

In this example two-phase gas-oil relative permeability is specified for one saturation region.

6.6.3   SOF2

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies relative permeability tables for two-phase water-oil systems for all saturation regions from TABDIMS (see 6.1.13). Each table is a 2-column table with the following columns:

- SO (oil saturation) (this is argument value for function below)

- KROW (oil permeability) (this is function k_{rOW} in 2.6)

Number of tables depends on number of saturation regions defined in TABDIMS. Each table must be terminated by a slash /.

Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SOF2
 0      0
 0.25  0
 0.3   0.002
 0.35  0.008
 0.4   0.018
 0.45  0.032
 0.5   0.05
 0.55  0.072
 0.6   0.098
 0.65  0.128
 0.7   0.162
 0.75  0.2
 1     0.2
 /
```

In this example two-phase oil-water relative permeability is specified for one saturation region.

6.6.4   SWFN

	tNavigator		eclipse 300		MORE Roxar
✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies relative permeability tables for all saturation function regions from TABDIMS (see 6.1.13) for two-phase water-oil systems. Each table is a 3-column table with the following columns:

- SW (water saturation) (this is argument value for functions below)

- KRWO (water permeability) (this is function k_{rWO} in 2.6)

- POW (oil-water capillary pressure) (this is function P_{cOW} in 2.12.2)

Number of tables depends on number of saturation function regions defined in TABDIMS. Each table must be terminated by a slash /.

Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SWFN
 0      0      0
 0.25  0      0
 0.3   0.002  0
 0.35  0.008  0
 0.4   0.018  0
 0.45  0.032  0
 0.5   0.05   0
 0.55  0.072  0
 0.6   0.098  0
 0.65  0.128  0
 0.7   0.162  0
 0.75  0.2    0
 1     0.2    0
 /
```

This example sets relative permeability for a two-phase water-oil system for one saturation function region.

6.6.5 SGFN

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies relative permeability tables for all saturation function regions from TABDIMS (see 6.1.13) for two-phase oil-gas systems. Each table is a 3-column table with the following columns:

- SG - gas saturation (this is argument value for functions below).

- KRGO - water permeability (this is function k_{rWO} in 2.6). First elemnt of this column must be zero.

- POG - oil-gas capillary pressure (this is function P_{cOW} in 2.12.1).

Number of tables depends on number of saturation function regions defined in TABDIMS. Each table must be terminated by a slash /.

Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SGFN
 0      0        0
 0.25   0        0
 0.3    0.002    0
 0.35   0.008    0
 0.4    0.018    0
 0.45   0.032    0
 0.5    0.05     0
 0.55   0.072    0
 0.6    0.098    0
 0.65   0.128    0
 0.7    0.162    0
 0.75   0.2      0
 1      0.2      0
 /
```

This example sets relative permeability for a two-phase oil-gas system for one saturation function region.

6.6.6   SOF3

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies relative permeability tables for three-phase systems for all saturation regions from TABDIMS (see 6.1.13). Each table should be terminated with a slash /.

Each 3-column table consists of the following columns:

1. **SO** (oil saturation) (this is argument value for functions below)

2. oil permeability (where only oil and water are present);

3. oil permeability (where oil, gas and water are present).

Default: values may be used in columns 2 and 3. These values will be computed by linear interpolation.

The keyword specifies relative permeability tables for two-phase systems – SOF2 (see 6.6.3).

```
Example
TABDIMS
1 1 2* 1
/
...
SOF3
 0      0      0
 0.25  0      0
 0.3   0.002  0
 0.35  0.008  1*
 0.4   0.018  0.01
 0.45  0.032  1*
 0.5   0.05   0.12
 0.55  0.072  1*
 0.6   0.098  0.3
 0.65  0.128  1*
 0.7   0.162  1*
 0.75  0.2    0.63
 1     0.2    1*
 /
```

This example sets relative permeability for three-phase system for one saturation function region.

6.6.7 SGWFN

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
✓	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies relative permeability tables for two-phase gas-water systems for all saturation regions from TABDIMS. Each table is a 4-column table with the following columns:

- SG (gas saturation) (this is argument value for functions below)

- KRGW (gas permeability)

- KRWG (water permeability)

- PGW (gas-water capillary pressure)

Number of tables depends on number of saturation regions defined in TABDIMS. Each table must be terminated by a slash /.

Default: none

```
Example
TABDIMS
1 1 2* 1
/
...
SGWFN
 0.27 0     1     0
 0.3  0.02  0.81  0.3
 0.42 0.18  0.49  0.9
 0.51 0.34  0.16  1.4
 0.6  0.53  0.09  2.7
 0.64 0.71  0.04  4.1
 0.72 0.92  0.01  5.3
 0.85 1.00  0     7.1
 1    1.00  0     9.4
 /
```

In this example two-phase gas-water relative permeability is specified for one saturation region.

6.6.8   TOLCRIT

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the tolerance for critical saturations, so the method for determination of the initial relative permeability critical tables saturations is modified. One value of tolerance should be specified. The data should be terminated with a slash /.

The keyword can be used in case of end-point scaling (ENDSCALE (see 6.6.11)) (2.6.3).

If TOLCRIT (see 6.6.8) isn't specified the critical water saturation S_{Wcr} is equal to S_W in the last table entry (SWOF (see 6.6.1), SWFN (see 6.6.4)) for k_{rW}, for which $k_{rW} \leq 1.0 * 10^{-6}$ – in e100 models ($k_{rW} \leq 1.0 * 10^{-20}$ – in e300 models) (finding the last zero relative permeability value while accounting for machine zero).

If TOLCRIT (see 6.6.8) is specified, the critical water saturation is equal to S_W in the last table entry, for $k_{rW} \leq TOLCRIT$ (analogously S_{Gcr}, S_{OWcr}, S_{OGcr}).

Defauls: $1.0 * 10^{-6}$ – in e100 models, $1.0 * 10^{-20}$ – in e300 models.

```
Example
TOLCRIT
1.0E-9 /
```

6.6.9   STONE1

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies that Stone 1 model will be used for oil relative permeability calculation. User should specify either STONE1, or STONE2 (see 6.6.10). If none is specified, the first Stone's model will be used, see model description in section 2.6.1.

Example
```
STONE1
```

6.6.10 STONE2

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies that Stone 2 model will be used for oil relative permeability calculation. User should specify either STONE1 (see 6.6.9), or STONE2. If none is specified, the first Stone's model will be used, see model descriptions in sections 2.6.1, 2.6.2.

Example
```
STONE2
```

6.6.11   ENDSCALE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

Indicates that end-point scaling of relative permeabilities and capillary pressures will be used, see 2.6.3, 2.6.4, 2.12.3 for details. Table end-points can then be entered cell by cell (SWL (see 6.6.14), SWCR (see 6.6.16), SWU (see 6.6.20), KRW (see 6.6.26), PCW (see 6.6.28)) or with respect to depth (ENPTVD (see 6.6.22), ENKRVD (see 6.6.23), ENPCVD (see 6.6.24)).

Only not directional reversible end-point scaling is supported.

The following parameters should be specified (The data should be terminated with a slash /.):

1. directional end-point scaling switch (DIRECT, NODIR), ignored, this is an Eclipse compatibility field;

2. irreversible end-point scaling switch (IRREVERS, REVERS), ignored, this is an Eclipse compatibility field;

3. the number of saturation end-point versus depth tables (keywords ENPTVD (see 6.6.22), ENKRVD (see 6.6.23)); in tNavigator number of tables does not have to be specified because the memory is allocated dynamically;

4. the maximum number of nodes in any saturation end-point versus depth table (keywords ENPTVD (see 6.6.22), ENKRVD (see 6.6.23)); in tNavigator number of tables does not have to be specified because the memory is allocated dynamically.

Default:

- the number of saturation end-point versus depth tables (keywords ENPTVD (see 6.6.22), ENKRVD (see 6.6.23)) – 1;

- the maximum number of nodes in any saturation end-point versus depth table (keywords ENPTVD (see 6.6.22), ENKRVD (see 6.6.23)) – 20.

Example
```
ENDSCALE
```

6.6.12 TZONE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword controls the transition zone option. The keyword can be used only is the keyword ENDSCALE (see 6.6.11) is active – indicates that end-point scaling of relative permeabilities and capillary pressures will be used.

3 parameters should be specified (The data should be terminated with a slash /.) (T – true, F – false). Each parameter corresponds to one phase: oil, water and gas.

If the parameter is set – true to a phase, then the critical saturations for that phase will be modified to be the initial immobile saturation in regions where the saturation is below the input critical value.

1. the parameter is set true to oil phase – SOWCR (see 6.6.18) will be modified for oil-water runs or oil-water-miscible gas runs, SOGCR (see 6.6.19) will be modified only for oil-gas runs;

2. the parameter is set true to water phase, SWCR (see 6.6.16) will be modified;

3. the parameter is set true to gas phase, SGCR (see 6.6.17) will be modified for gas-water runs and oil-gas runs.

Example
```
TZONE
F T T /
```

6.6.13 SCALECRS

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Keyword may be present in data file, if ENDSCALE (see 6.6.11) is selected. Allows to define three-point scaling method for saturation tables end-point scaling. See 2.6.3, 2.6.4 for details.

Keyword takes one argument, with two possible values

- YES (three-point scaling method) or NO (two-point scaling method)

Next line must contain a slash /.

Default: NO

Example
```
SCALECRS
YES
/
```

This example sets three-point scaling mehtod.

6.6.14 SWL

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines minimal water saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: connate water saturation in corresponding saturation region, i.e. minimal water saturation in SWOF (see 6.6.1) table.

```
Example
DIMENS
5 5 4
/
...
SWL
50*0.35 50*0.45
/
```

This example defines connate water saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.15   SGL

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines minimal gas saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: connate gas saturation in corresponding saturation region, i.e. minimal gas saturation in SGOF (see 6.6.2) table.

```
Example
DIMENS
5 5 4
/
...
SGL
50*0.35 50*0.45
/
```

This example defines connate gas saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.16 SWCR

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines critical water saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: critical water saturations in corresponding saturation region, i.e. maximal water saturation in SWOF (see 6.6.1), for which $k_{rw} = 0$.

```
Example
DIMENS
5 5 4
/
...
SWCR
50*0.35 50*0.45
/
```

This example defines critical water saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.17 SGCR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines critical gas saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: critical gas saturations in corresponding saturation region, i.e. maximal gas saturation in SGOF (see 6.6.2), for which $k_{rG} = 0$.

```
Example
DIMENS
5 5 4
/
...
SGCR
50*0.35 50*0.45
/
```

This example defines critical gas saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.18 SOWCR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines critical oil-to-water saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if **ENDSCALE** (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: critical oil-to-water saturation in corresponding saturation region, i.e. maximal oil saturation in **SWOF** (see 6.6.1), for which the oil relative permeability is zero: $k_{rOW} = 0$.

```
Example
DIMENS
5 5 4
/
...
SOWCR
50*0.35 50*0.45
/
```

This example defines critical oil-to-water saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.19 SOGCR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines critical oil-to-gas saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: critical oil-to-gas saturation in corresponding saturation region, i.e. maximal oil saturation in SGOF (see 6.6.2), for which the oil relative permeability is zero: $k_{rOG} = 0$.

```
Example
DIMENS
5 5 4
/
...
SOGCR
50*0.35 50*0.45
/
```

This example defines critical oil-to-gas saturation = 0.35 for first 50 blocks of the grid and 0.45 for last 50 blocks.

6.6.20 SWU

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines maximal water saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: maximal water saturation in corresponding saturation region, i.e. maximal value of water saturation in SWOF (see 6.6.1).

```
Example
DIMENS
5 5 4
/
...
SWU
50*0.75 50*0.85
/
```

This example defines maximal water saturation = 0.75 for first 50 blocks of the grid and 0.85 for last 50 blocks.

6.6.21   SGU

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword defines maximal gas saturation for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4, 2.12.3 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: maximal gas saturation in corresponding saturation region, i.e. maximal value of gas saturation in SGOF (see 6.6.2).

```
Example
DIMENS
5 5 4
/
...
SGU
50*0.75 50*0.85
/
```

This example defines maximal gas saturation = 0.75 for first 50 blocks of the grid and 0.85 for last 50 blocks.

6.6.22   ENPTVD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies saturation end point versus depth tables. The number of tables should not be greater then 3-rd parameter of the keyword ENDSCALE (see 6.6.11). Each table should be terminated with a slash /.

One table row should contain the following parameters:

1. depth (m);

2. the corresponding values of the connate water saturation (if table end-points are entered cell by cell – the keyword SWL (see 6.6.14) can be used);

3. the corresponding values of the critical water saturation (if table end-points are entered cell by cell – the keyword SWCR (see 6.6.16) can be used);

4. the corresponding values of the maximum water saturation (if table end-points are entered cell by cell – the keyword SWU (see 6.6.20) can be used);

5. the corresponding values of the connate gas saturation (if table end-points are entered cell by cell – the keyword SGL (see 6.6.15) can be used);

6. the corresponding values of the critical gas saturation (if table end-points are entered cell by cell – the keyword SGCR (see 6.6.17) can be used);

7. the corresponding values of the maximum gas saturation (if table end-points are entered cell by cell – the keyword SGU (see 6.6.21) can be used);

8. the corresponding values of the critical oil-in-water saturation (if table end-points are entered cell by cell – the keyword SOWCR (see 6.6.18) can be used);

9. the corresponding values of the critical oil-in-gas saturation (if table end-points are entered cell by cell – the keyword SOGCR (see 6.6.19) can be used).

Each saturation entry (parameters 2-9) should be in the range 0.0 to 1.0 inclusive.

```
Example
ENDSCALE
2* 1 2 /
...
ENPTVD
3500.0 0.20 0.20 1.0 0.0 0.04 1.0 0.18 0.22
7500.0 0.22 0.22 1.0 0.0 0.04 1.0 0.18 0.22 /
```

In this example one table is specified.

6.6.23   ENKRVD

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies relative permeability end point versus depth tables. The number of tables should not be greater then 3-rd parameter of the keyword ENDSCALE (see 6.6.11). Each table should be terminated with a slash /.

One table row should contain the following parameters:

1. depth (m);

2. the corresponding values of the maximum water relative permeability (if relative permeabilities are entered cell by cell – the keyword KRW (see 6.6.26) can be used);

3. the corresponding values of the maximum gas relative permeability (if relative permeabilities are entered cell by cell – the keyword KRG (see 6.6.27) can be used);

4. the corresponding values of the maximum oil relative permeability (if relative permeabilities are entered cell by cell – the keyword KRO (see 6.6.25) can be used);

5. water relative permeability at the critical oil (or gas) saturation (if relative permeabilities are entered cell by cell – the keyword KRWR (see 6.6.26) can be used);

6. gas relative permeability at the critical oil (or water) saturation (if relative permeabilities are entered cell by cell – the keyword KRGR (see 6.6.27) can be used);

7. oil relative permeability at the critical gas saturation (if relative permeabilities are entered cell by cell – the keyword KRORG (see 6.6.25) can be used);

8. oil relative permeability at the critical water saturation (if relative permeabilities are entered cell by cell – the keyword KRORW (see 6.6.25) can be used).

Each permeability entry (parameters 2-8) should be in the range 0.0 to 1.0 inclusive.

```
Example
ENDSCALE
2* 1 2 /
...
ENKRVD
3000.0 0.50 1.0 1.0 0.3 1* 1* 0.34
6000.0 0.58 1.0 1.0 1* 1* 1* 0.36
9000.0 0.66 0.95 0.95 0.35 1* 1* 0.37/
```

In this example one table is specified.

6.6.24 ENPCVD

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies maximum capillary pressure versus depth tables. The number of tables should not be greater then 3-rd parameter of the keyword ENDSCALE (see 6.6.11). Each table should be terminated with a slash /.

One table row should contain the following parameters:

1. depth (m);

2. the corresponding values of the maximum gas-oil capillary pressure (if capillary pressures are entered cell by cell – the keyword PCG (see 6.6.29) can be used) (*bars*);

3. the corresponding values of the maximum water-oil capillary pressure (if capillary pressures are entered cell by cell – the keyword PCW (see 6.6.28) can be used) (*bars*).

```
Example
ENDSCALE
2* 1 2 /
...
ENPCVD
2500.0 1.6 22.0
7500.0 1.3 31.0 /
```

In this example one table is specified.

6.6.25   KRO, KRORW, KRORG

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

These keywords define maximal oil relative permeability at different saturations for grid blocks, used for saturation end point scaling. Keywords may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: value of relative permeability from table for corresponding saturation.

```
Example
DIMENS
5 5 4
/
...
KRO
50*0.9 50*1.
/
KRORW
50*0.5 50*0.4
/
```

This example defines maximal relative permeability to oil as 0.9 for first 50 blocks of the grid and 1 for last 50 blocks. Relative permeability at critical saturation of displacing phase is equal to 0.5 and 0.4 correspondingly.

6.6.26 KRW, KRWR

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

These keywords define maximal water relative permeability at different saturations for grid blocks, used for saturation end point scaling. Keywords may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: value of relative permeability from table for corresponding saturation.

```
Example
DIMENS
5 5 4
/
...
KRW
50*0.9 50*1.
/
KRWR
50*0.5 50*0.4
/
```

This example defines maximal relative permeability to water as 0.9 for first 50 blocks of the grid and 1 for last 50 blocks. Relative permeability at critical saturation of displacing phase is equal to 0.5 and 0.4 correspondingly.

6.6.27　KRG, KRGR

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

These keywords define maximal gas relative permeability at different saturations for grid blocks, used for saturation end point scaling. Keywords may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.6.3, 2.6.4 for details of end-point scaling.

The same number of values as number of cells must be specified.

Default: value of relative permeability from table for corresponding saturation.

```
Example
DIMENS
5 5 4
/
...
KRG
50*0.9 50*1.
/
KRGR
50*0.5 50*0.4
/
```

This example defines maximal relative permeability to gas as 0.9 for first 50 blocks of the grid and 1 for last 50 blocks. Relative permeability at critical saturation of displacing phase is equal to 0.5 and 0.4 correspondingly.

6.6.28   PCW

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword defines maximal oil-water capillary pressure for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.12.3 for details.

The same number of values as number of cells must be specified.

Default: maximal oil-water capillary pressure in corresponding saturation region, i.e. maximum capillary pressure from table SWOF (see 6.6.1).

```
Example
DIMENS
5 5 4
/
...
PCW
50*5 50*3
/
```

This example defines maximal oil-water capillary pressure = 5 for first 50 blocks of the grid and 3 for last 50 blocks.

6.6.29   PCG

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines maximal oil-gas capillary pressure for grid blocks, used for saturation end point scaling. Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file. See 2.12.3 for details.

The same number of values as number of cells must be specified.

Default: maximal oil-gas capillary pressure in corresponding saturation region, i.e. maximum capillary pressure from table SGOF (see 6.6.2).

Example
```
DIMENS
5 5 4
/
...
PCG
50*5 50*3
/
```

This example defines maximal oil-gas capillary pressure = 5 for first 50 blocks of the grid and 3 for last 50 blocks.

6.6.30 SWATINIT

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines initial water saturation for grid blocks, set up using saturation end point scaling. Water capillary pressure will be scaled in order to make equilibrium water saturation equal to indicated one (this means that EQUIL (see 6.9.1) should be used for initialization). When SWATINIT is used, maximum capillary pressure array PCW is generated (see 2.12.3 for details). Water saturation set up by SWATINIT will not always be honoured, especially for the case when division by zero should be performed to scale pressure (see notes).

Keyword may be used only if ENDSCALE (see 6.6.11) is present in data file.

The same number of values as number of cells must be specified.

Notes:

- For grid blocks below oil water contact, where capillary pressure is zero, if SWATINIT saturation is set below 1.0, it will not be honoured.

- For grid blocks above oil water contact, if SWATINIT saturation is set to value corresponding to zero P_{cw} (usually it's $S_W = 1.0$), it will not be honoured.

- If SWATINIT saturation is less or equal to connate water saturation, it will not be honoured.

- If 9th argument in EQUIL (see 6.9.1) is not set to zero, SWATINIT saturation will not exactly be honoured due to fine layer equilibrium calculation.

- If PCW (see 6.6.28) keyword is set, it's values will be ignored.

- If JFUNC (see 6.2.47) keyword is used, maximum capillary pressures P_{cw} calculated from it will be ignored.

```
Example
DIMENS
5 5 4
/
...
SWATINIT
50*0.05 25*0.3 25*1
/
```

This example defines initial water saturation = 0.05 for first 50 blocks of the grid, 0.3 for next layer, 1 for bottom layer.

6.7   Compositional properties

This section describes reservoir fluid properties for a compositional run. The keywords listed in this section fully describe three-phase multi-component non-isothermic model.

For water and rock property definition keywords from previous section are used (PVTW (see 6.5.4), ROCK (see 6.5.12), DENSITY (see 6.5.15), SWOF (see 6.6.1), 6.6.2).

For some components, listed below, default values of parameters may be used (TCRIT (see 6.7.11), PCRIT (see 6.7.12), VCRIT (see 6.7.13), MW (see 6.7.14), BIC (see 6.7.16)). In this case component properties will be taken from the following table (in compliance with their names indicated in CNAMES (see 6.7.4)).

6.7.1   Default values of component properties for compositional model. Part 1

Component name	Chemical formula	Critical temperature, K	Critical pressure, BARSA	Acentric factor	Molecular weight	Critical volume
N2	N_2	$1.2620E+02$	$3.394387500E+01$	$4.000000000E-02$	$2.8013000E+01$	$9.00E-02$
CO2	CO_2	$3.0470E+02$	$7.386592500E+01$	$2.250000000E-01$	$4.4010000E+01$	$9.40E-02$
H2S	H_2S	$3.7360E+02$	$8.936865000E+01$	$1.000000000E-01$	$3.4076000E+01$	$9.80E-02$
C1	C_1H_3	$1.9060E+02$	$4.604208000E+01$	$1.300000000E-02$	$1.6043000E+01$	$9.80E-02$
C2	C_2H_6	$3.0543E+02$	$4.883865000E+01$	$9.860000000E-02$	$3.0070000E+01$	$1.48E-01$
C3	C_3H_8	$3.6980E+02$	$4.245517500E+01$	$1.524000000E-01$	$4.4097000E+01$	$2.00E-01$
IC4	iC_4H_{10}	$4.0810E+02$	$3.647700000E+01$	$1.848000000E-01$	$5.8123999E+01$	$2.63E-01$
NC4	nC_4H_{10}	$4.2520E+02$	$3.796647750E+01$	$2.010000000E-01$	$5.8124001E+01$	$2.55E-01$
IC5	iC_5H_{12}	$4.6040E+02$	$3.389321250E+01$	$2.270000000E-01$	$7.2150999E+01$	$3.08E-01$
NC5	nC_5H_{12}	$4.6960E+02$	$3.370069500E+01$	$2.510000000E-01$	$7.2151001E+01$	$3.11E-01$
C6	C_6H_{14}	$5.0750E+02$	$3.010365750E+01$	$2.990000000E-01$	$8.6178000E+01$	$3.51E-01$
C7	C_7H_{16}	$5.4800E+02$	$2.938425000E+01$	$3.000000000E-01$	$9.6000000E+01$	$3.92E-01$
C8	C_8H_{18}	$5.7500E+02$	$2.879656500E+01$	$3.120000000E-01$	$1.0700000E+02$	$4.33E-01$
C9	C_9H_{20}	$6.0300E+02$	$2.630397000E+01$	$3.480000000E-01$	$1.2100000E+02$	$4.84E-01$
C10	$C_{10}H_{22}$	$6.2600E+02$	$2.419641000E+01$	$3.850000000E-01$	$1.3400000E+02$	$5.34E-01$
C11	$C_{11}H_{24}$	$6.4800E+02$	$2.230163273E+01$	$4.189999998E-01$	$1.4700000E+02$	$5.87E-01$
C12	$C_{12}H_{26}$	$6.6800E+02$	$2.080202250E+01$	$4.539999962E-01$	$1.6100000E+02$	$6.37E-01$
C13	$C_{13}H_{28}$	$6.8700E+02$	$1.959625500E+01$	$4.839999974E-01$	$1.7500000E+02$	$6.83E-01$
C14	$C_{14}H_{30}$	$7.0600E+02$	$1.860327000E+01$	$5.159999728E-01$	$1.9000000E+02$	$7.30E-01$
C15	$C_{15}H_{32}$	$7.2400E+02$	$1.760015250E+01$	$5.500000119E-01$	$2.0600000E+02$	$7.79E-01$
C16	$C_{16}H_{34}$	$7.4000E+02$	$1.659703500E+01$	$5.820000172E-01$	$2.2200000E+02$	$8.31E-01$
C17	$C_{17}H_{36}$	$7.5500E+02$	$1.589789250E+01$	$6.129997757E-01$	$2.3700000E+02$	$8.74E-01$
C18	$C_{18}H_{38}$	$7.6700E+02$	$1.530007500E+01$	$6.380001114E-01$	$2.5100000E+02$	$9.13E-01$
C19	$C_{19}H_{40}$	$7.7800E+02$	$1.480358250E+01$	$6.620000005E-01$	$2.6300000E+02$	$9.49E-01$
C20	$C_{20}H_{42}$	$7.9000E+02$	$1.433748750E+01$	$6.899999976E-01$	$2.7500000E+02$	$9.94E-01$

6.7.2   Default values of component properties for compositional model. Part 2: Binary interaction coefficients

	N2	CO2	H2S	C1	C2
N2					
CO2	$-1.200000010E-02$				
H2S	$1.000000015E-01$	$1.000000015E-01$			
C1	$1.000000015E-01$	$1.000000015E-01$	0		
C2	$1.000000015E-01$	$1.000000015E-01$	0	0	
C3	$1.000000015E-01$	$1.000000015E-01$	0	0	0
IC4	$1.000000015E-01$	$1.000000015E-01$	0	0	0
NC4	$1.000000015E-01$	$1.000000015E-01$	0	0	0
IC5	$1.000000015E-01$	$1.000000015E-01$	0	0	0
NC5	$1.000000015E-01$	$1.000000015E-01$	$2.873999672E-02$	$9.999999776E-03$	$9.999999776E-03$
C6	$1.000000015E-01$	$1.000000015E-01$	$3.391999674E-02$	$9.999999776E-03$	$9.999999776E-03$
C7	$1.000000015E-01$	$1.000000015E-01$	$3.699999675E-02$	$9.999999776E-03$	$9.999999776E-03$
C8	$1.000000015E-01$	$1.000000015E-01$	$3.965999676E-02$	$9.999999776E-03$	$9.999999776E-03$
C9	$1.000000015E-01$	$1.000000015E-01$	$4.161999677E-02$	$9.999999776E-03$	$9.999999776E-03$
C10	$1.000000015E-01$	$1.000000015E-01$	$4.315999678E-02$	$9.999999776E-03$	$9.999999776E-03$
C11	$1.000000015E-01$	$1.000000015E-01$	$4.469999678E-02$	$9.999999776E-03$	$9.999999776E-03$
C12	$1.000000015E-01$	$1.000000015E-01$	$4.623999679E-02$	$9.999999776E-03$	$9.999999776E-03$
C13	$1.000000015E-01$	$1.000000015E-01$	$4.777999680E-02$	$9.999999776E-03$	$9.999999776E-03$
C14	$1.000000015E-01$	$1.000000015E-01$	$4.917999680E-02$	$9.999999776E-03$	$9.999999776E-03$
C15	$1.000000015E-01$	$1.000000015E-01$	$5.015999681E-02$	$9.999999776E-03$	$9.999999776E-03$
C16	$1.000000015E-01$	$1.000000015E-01$	$5.127999681E-02$	$9.999999776E-03$	$9.999999776E-03$
C17	$1.000000015E-01$	$1.000000015E-01$	$5.197999682E-02$	$9.999999776E-03$	$9.999999776E-03$
C18	$1.000000015E-01$	$1.000000015E-01$	$5.267999682E-02$	$9.999999776E-03$	$9.999999776E-03$
C19	$1.000000015E-01$	$1.000000015E-01$	$5.337999682E-02$	$9.999999776E-03$	$9.999999776E-03$
C20	0	0	0	0	0

The rest coefficients of lower triangular part of binary interaction matrix are equal to zero by default.

6.7.3   COMPS

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword sets the number of components in a compositional run – N. Presence of this keyword indicates that the compositional mode should be turned on.

Default: none

```
Example
COMPS
6
/
```

In this example compositional mode is activated. The oil-gas mixture will consist of 6 components.

6.7.4 CNAMES

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword introduces component names in a compositional run. If component name coincides with name from table 6.7.1 on page 342, some parameter values for this component may be defaulted (such as critical temperature (T_c), critical pressure (p_c) etc.).

Default: none

```
Example
COMPS
5
/
...
CNAMES
CO2 C1 NC4 IC4 C10
/
```

In this example oil-gas mixture consists of 5 components: carbon dioxide (CO_2), methane (CH_4), n-butane (nC_4H_{10}), iso-butane (iC_4H_{10}) and decane ($C_{10}H_{22}$).

6.7.5 EOS

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword specifies for each EoS region which equation of state should be used for liquid-vapor equilibrium calculation. Current simulator version supports three types of cubic equations of state in the form of Redlich-Kwong (RK), Soave-Redlich-Kwong (SRK) and Peng-Robinson (PR).

Equation name for each region starts from new line, each line must be terminated by a slash /. Number of strings depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

Default: PR

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
EOS
SRK
/
```

Here Soave-Redlich-Kwong equation of state will be used for liquid-vapor equilibrium calculation.

```
Example
TABDIMS
2 3 2* 4 4*3
/
...
EOS
SRK
/
RK
/
PR
/
```

Here in each of 3 EoS regions it's own type of equation of state will be used.

6.7.6 RTEMP

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets constant temperature of each EoS region in case when isothermic compositional model is used. In case of metric system temperature should be speicifed in Celcium degrees, for field units in Fahrenheit degrees.

Temperature value for each region starts from new line, each line must be terminated by a slash /. Number of strings depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

Default: none

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
RTEMP
80
/
```

Here isothermic compositional reservoir model is used, reservoir temperature is 80 Celcium degrees.

```
Example
TABDIMS
2 3 2* 4 4*3
/
...
RTEMP
90
/
80
/
90
/
```

In this example each of three EoS regions will have it's own constant temperature.

6.7.7 STCOND

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets standard pressure and temperature for calculation of gas, oil, water volume in standard conditions. User should specify two values: standard temperature and pressure.

Default: values from 4

```
Example
STCOND
20 2
/
```

6.7.8 XMFVP

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies the table of liquid composition versus pressure (distribution of concentration matrix $x_{c,O}$ $c = 1, \ldots, n'_c$. The data should be terminated with a slash /.
The table should be specified for each equiblium region (EQLDIMS (see 6.1.14)).

The numbers of columns depends on the number of components (COMPS (see 6.7.3)).

One table row contains the following parameters:

- pressure P;

- the corresponding value of concentration $x_{1,O}$ of first component at pressure P;

- the corresponding value of concentration $x_{2,O}$ of second component at pressure P;

- ...

- the corresponding value of concentration $x_{n'_c,O}$ of n'_c component at pressure P

An arbitrary number of lines with different pressures can be specified. The data should be terminated with a slash /.

```
Example
XMFVP
1200 0.025 0.025 0.14 0.65 0.16
1800 0.025 0.025 0.14 0.65 0.16
/
```

In this example consentration matrix is specified for two different values of pressure.

6.7.9 YMFVP

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword specifies the table of vapor composition versus pressure (distribution of concentration matrix $x_{c,G}$ $c = 1, \ldots, n'_c$, $P = O, G$ (для $P = W$)). The data should be terminated with a slash /.
The table should be specified for each equiliblium region (EQLDIMS (see 6.1.14)).

The numbers of columns depends on the number of components (COMPS (see 6.7.3)).

One table row contains the following parameters:

- pressure P;

- the corresponding value of concentration $x_{1,G}$ of first component at pressure P;

- the corresponding value of concentration $x_{2,G}$ of second component at pressure P;

- ...

- the corresponding value of concentration $x_{n'_c,G}$ of n'_c component at pressure P

An arbitrary number of lines with different pressures can be specified. The data should be terminated with a slash /.

Example
```
YMFVP
1200 0.025 0.025 0.14 0.65 0.16
1800 0.025 0.025 0.14 0.65 0.16
/
```

In this example consentration matrix is specified for two different values of pressure.

6.7.10 ZMFVD

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword sets initial composition with respect to depth data for each equilibration region (for a compositional run). First column lists depth values, the rest N columns (N is number of components defined in COMPS, see 6.7.3) list molar composition of corresponding components for this depth. Note that molar fractions on each line must add up to unity.

For depth values outside the specified range, constant value extrapolation will be performed. Table may consist of only one line; this would result in constant component distribution for all equilibration regions.

Number of tables depends on number of equilibration regions defined in EQLDIMS (see 6.1.14). Each table must be terminated by a slash /.

Default: none

```
Example
EQLDIMS
1 /
...
COMPS
5
/
...
ZMFVD
1800 0.01 0.64 0.19 0.06 0.1
2000 0.01 0.62 0.18 0.07 0.12
/
```

In this example mixture composition is defined for depths 1800 and 2000. Between these depths linear interpolation will be used, outside the interval composition will remain constant.

```
Example
EQLDIMS
2 /
...
COMPS
5
/
...
ZMFVD
1800 0.01 0.64 0.19 0.06 0.1
2000 0.01 0.62 0.18 0.07 0.12
/
2000 0.01 0.62 0.18 0.07 0.12
/
```

In this example mixture composition in first equilibration region is defined for depths 1800 and 2000. Between these depths linear interpolation will be used, outside the interval composition will remain constant. In the second equilibration region constant composition is defined.

6.7.11   TCRIT

✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword associates critical temperature with each component of a compositional model for each EoS region. For each region a line of N values should be entered, where N is number of components defined in COMPS (see 6.7.3). Each line must be terminated by a slash /. Number of lines depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

In case of metric system temperature should be specified in Kelvin degrees, and for field unit system in Rankine degrees (see UNITS, 6.1.10).

If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.1

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
TCRIT
3.047000000E+02 1* 4.252000000E+02 6.260000000E+02
/
```

Here critical temperature is specified for components $\{CO_2, C_1, NC_4, C_{10}\}$. Value for component C_1 will be defaulted from table 6.7.1.

Example
```
TABDIMS
1 2 2* 4 4*2
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C8
/
...
TCRIT
4*
/
3* 6.260000000E+02
/
```

Here critical temperature is specified for components $\{CO_2, C_1, NC_4, C_8\}$. In the first region default temperature values will be taken from table 6.7.1, in the second region all values but for component C_8 will also be taken by default.

6.7.12 PCRIT

√	tNavigator	√ eclipse 300	☐ MORE Roxar
☐	eclipse 100	☐ CMG imex	☐ CMG stars

The keyword associates critical pressure with each component of a compositional model for each EoS region. For each region a line of N values should be entered, where N is number of components defined in COMPS (see 6.7.3). Each line must be terminated by a slash /. Number of lines depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.1

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
PCRIT
1* 4.604208000E+01 3.796647750E+01 2.419641000E+01
/
```

Here critical pressure for components $\{CO_2, C_1, NC_4, C_{10}\}$ is specified. Value for CO_2 will be defaulted from table 6.7.1.

Example
```
TABDIMS
2 2 2* 4 4*2
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C8
/
...
PCRIT
7.386592500E+01 4.604208000E+01 3.796647750E+01 2.87000E+01
/
1* 4.604209000E+01 2*
/
```

Here critical pressure for components $\{CO_2, C_1, NC_4, C_8\}$ is specified. In the second region all values but for component C_1 will be taken by default from table 6.7.1.

6.7.13   VCRIT

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword associates critical volume with each component of a compositional model for each EoS region. For each region a line of N values should be entered, where N is number of components defined in COMPS (see 6.7.3). Each line must be terminated by a slash /. Number of lines depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.1

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
VCRIT
3* 5.340000000E-01

/
```

Here critical volumes for components $\{CO_2, C_1, NC_4, C_{10}\}$ are specified. Values for components CO_2, C_1, NC_4 are taken by default from table 6.7.1.

Example
```
TABDIMS
2 2 2* 4 4*2
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
VCRIT
3* 5.340000000E-01
/
3* 5.350000000E-01
/
```

Here critical volumes for components $\{CO_2, C_1, NC_4, C_{10}\}$ are specified for two regions. Values for components CO_2, C_1, NC_4 will be taken by default from table 6.7.1.

6.7.14 MW

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword associates molecular weight with each component of a compositional model for each EoS region. For each region a line of N values should be entered, where N is number of components defined in COMPS (see 6.7.3). Each line must be terminated by a slash /. Number of lines depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.1

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
MW
4.401000000E+01 1.604300000E+01 5.812400000E+01
1.340000000E+02
/
```

Here molecular weights for components $\{CO_2, C_1, NC_4, C_{10}\}$ are specified.

Example
```
TABDIMS
2 3 2* 4 4*3
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
MW
3* 1.342000000E+02
/
3* 1.343000000E+02
/
4.401000000E+01 1.604300000E+01 5.812400000E+01
1.342000000E+02
/
```

Here molecular weights for components $\{CO_2, C_1, NC_4, C_{10}\}$ are specified in three EoS regions. Molecular weights of first three components in first two regions will be taken by default from table 6.7.1.

6.7.15 ACF

√	**tNavigator**	√	**eclipse 300**		**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword associates acentric factor with each component of a compositional model for each EoS region. For each region a line of N values should be entered, where N is number of components defined in COMPS (see 6.7.3). Each line must be terminated by a slash /. Number of lines depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter).

If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.1

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
ACF
2.250000000E-01 1.300000000E-02 1* 3.850000000E-01
/
```

Here acentric factors are specified for components $\{CO_2, C_1, NC_4, C_{10}\}$. Value for NC_4 will be taken by default from table 6.7.1.

Example
```
TABDIMS
2 2 2* 4 4*2
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C8
/
...
ACF
3* 0.311
/
2.250000000E-01 1.300000000E-02 1* 0.313
/
```

Here acentric factors are specified for components $\{CO_2, C_1, NC_4, C_8\}$. Values for components $\{CO_2, C_1, NC_4\}$ in the first region and for component NC_4 in the second region will be taken by default from table 6.7.1.

6.7.16 BIC

	tNavigator		eclipse 300		MORE Roxar
√	tNavigator	√	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies binary interaction coefficients between the components of a compositional model for each EoS region. For each region a table of $N(N-1)/2$ values should be specified, where N is number of components defined in COMPS (see 6.7.3). These elements form lower triangular part of binary interaction matrix.

Number of tables depends on number of EoS regions defined in TABDIMS (see 6.1.13) (9-th parameter). Each table must be terminated by a slash /.If component name is present in table 6.7.1, default values may be used for this component.

Default: values from table 6.7.2

```
Example
TABDIMS
2 1 2* 4 4*1
/
...
COMPS
4
/
CNAMES
CO2 C1 NC4 C10
/
...
BIC
1.000000015E-01
1.000000015E-01 0.000000000E+00
1.000000015E-01 4.161999677E-02 0.000000000E+00
/
```

Here binary interaction coefficients for components $\{CO_2, C_1, NC_4, C_{10}\}$ are specified.

```
Example
TABDIMS
2 2 2* 4 4*2
/
...
COMPS
3
/
CNAMES
methane ethane Cplus
/
...
BIC
-1.000000015E-01
1.000000015E-01 0.000000000E+00
/
-1.000000016E-01
1.000000015E-01 0.000000000E+00
/
```

Here binary interaction coefficients are specified for two EoS regions.

6.8 Thermal properties

6.8.1   CVTYPE

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the component volatility type for each component. N values should be entered (N – number of components COMPS (see 6.7.3)). The data should be terminated with a slash /.

The values should be the following:

- LIVE – if component is both volatile and condensable;

- SOLID – if component is only in solid phase;

- DEAD – if component is only in oil phase;

- GAS – if component is only in gas phase.

```
Example
COMPS
4 /
...
CVTYPE
LIVE GAS DEAD GAS /
```

6.8.2 WATDENT

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets water molar density as s function of temperature for each PVT region (the number of PVT regions is equal to the 2-nd parameter of TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

The following parameters are to be specified:

1. reference temperature $T_{w,ref}$ ($^\circ$K);

2. the first thermal expansion coefficient $c_{w,1,T}$ ($1/^\circ$K);

3. the second thermal expansion coefficient $c_{w,2,T}$ ($1/^\circ$K^2).

In e300 data format **the mass density** and **the molar water density** on default (or if the keyword THANALB (see 6.8.3) is present) are calculated

$$\rho_W = \frac{A_0 + A_1 T + A_2 T^2 + A_3 T^3 + A_4 T^4 + A_5 T^5}{1 + A_6 T} e^{c_{w,p}(p - A_7)}, \quad \xi_W = \frac{1}{MW_W} \cdot \rho_W,$$

where

$$\begin{aligned}
A_0 &= 9998.3952 & A_1 &= 169.55176 \\
A_2 &= -7.987 \times 10^{-2} & A_3 &= -46.170461 \times 10^{-5} \\
A_4 &= 105.56302 \times 10^{-8} & A_5 &= -280.54353 \times 10^{-11} \\
A_6 &= 16.87985 \times 10^{-2} & A_7 &= -102
\end{aligned}$$

If the keyword WATDENT is enable, then

$$\rho_W = \frac{\rho_{w,ref}}{(1 - c_{w,p}(p - p_{w,ref}))(1 + c_{w,1,T}(T - T_{w,ref}) + c_{w,2,T}(T - T_{w,ref})^2)}, \quad \xi_W = \frac{1}{MW_W} \cdot \rho_W$$

(6.1)

where

- $\rho_{w,ref} = \rho_{W,SC}/B_W(p_{ref})$ (kg/m^3)

- $\rho_{W,SC}$ – is set via DENSITY (see 6.5.15)

- $c_{w,p}$, $p_{w,ref}$, $B_W(p_{ref})$, – is set via PVTW (see 6.5.4)

- $c_{w,1,T}$, $c_{w,2,T}$, $T_{w,ref}$, – is set via WATDENT

- MW_W – water molar weight

In `stars` data format **the mass density** and **the molar water density** are calculated

$$\xi_W = \rho_{w,ref} \exp\left(c_{w,p}(p - p_{ref}) - c_{w,1,T}(T - T_{ref}) - c_{w,2,T}\frac{T^2 - T_{ref}^2}{2} + c_{w,pT}(p - p_{ref})(T - T_{ref})\right)$$

$$\rho_W = \xi_W \cdot MW_W$$

where

- p_{ref} – reference pressure PRSR (see 8.4.3)

- T_{ref} – reference temperature TEMR (see 8.4.4)

- $\rho_{w,ref}$ – плотность компоненты w из MOLDEN (see 8.4.5) (mol/m^3)

- $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – the properties of component w specified via CP (see 8.4.7), CT1 (see 8.4.8), CT2 (see 8.4.9), CPT (see 8.4.10)

Example
```
WATDENT
293 0.00027 0.0000034 /
```

6.8.3 THANALB

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets that water molar and mass densities are calculated using the following formulas

$$\rho_W = \frac{A_0 + A_1 T + A_2 T^2 + A_3 T^3 + A_4 T^4 + A_5 T^5}{1 + A_6 T} e^{c_{w,p}(p - A_7)}, \quad \xi_W = \frac{1}{MW_W} \cdot \rho_W,$$

где

$$A_0 = 9998.3952 \qquad A_1 = 169.55176$$
$$A_2 = -7.987 \times 10^{-2} \qquad A_3 = -46.170461 \times 10^{-5}$$
$$A_4 = 105.56302 \times 10^{-8} \qquad A_5 = -280.54353 \times 10^{-11}$$
$$A_6 = 16.87985 \times 10^{-2} \qquad A_7 = -102$$

If the keyword is disable but WATDENT (see 6.8.2) is specified, then densities are calculated via 3.6.

Example
`THANALB`

6.8.4 KVCR

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets 5 constants (A_i, B_i, C_i, D_i, E_i) for each component. These constants are used in the correlation formula:

$$K_i(p,T) = (A_i + B_i/p + C_i p) \cdot e^{-D_i/(T - E_i)}$$

where p – pressure, T – temperature, i – component number, $K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases.
The data should be terminated with a slash /.

The following parameters are to be specified:

1. A_i; N values should be entered (N – number of components COMPS (see 6.7.3)).

2. B_i; N values should be entered (N – number of components COMPS (see 6.7.3)).

3. C_i; N values should be entered (N – number of components COMPS (see 6.7.3)).

4. D_i; N values should be entered (N – number of components COMPS (see 6.7.3)).

5. E_i; N values should be entered (N – number of components COMPS (see 6.7.3)).

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

$K_i(p,T)$-values can be entered manually using the keywords KVTEMP (see 6.8.5), KVTABTn (see 6.8.6).

```
Example
COMPS
3 /
...
KVCR
300 800 523
236030 544000 345220
0 0 0
5000 11000 7000
220 0 0
/
```

This example sets the coefficients for the correlation formula for 3 components.

6.8.5   KVTEMP

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
☐	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the temperature values ($^\circ$C) for $K_i(p,T)$ tables entered using the keyword
KVTABTn (see 6.8.6) for each component. $K_i(p,T)$ – the ratio of mole fractions of component
in vapor and liquid phases. The number of temperature values should be equal to the 15-th
parameter of TABDIMS (see 6.1.13). The data should be terminated with a slash /.

$K_i(p,T)$-values can be entered using the correlation formula (KVCR (see 6.8.4)).

Example
```
KVTEMP
300 500 700
/
```

This example sets 3 temperature values for $K_i(p,T)$ tables.

6.8.6 KVTABTn

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets $K_i(p,T)$ tables for each component. $K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases. Each KVTABTn (see 6.8.6) corresponds to its temperature, entered using KVTEMP (see 6.8.5).

The following parameters should be specified in one data line:

- the oil phase pressure;

- N values of $K_i(p,T)$ at this pressure (N - number of components COMPS (see 6.7.3)).

An arbitrary number of keywords can be entered. The data should be terminated with a slash /.

$K_i(p,T)$-values can be entered using the correlation formula (KVCR (see 6.8.4)).

```
Example
KVTEMP
300 500 /
KVTABT1
45 0.0155 0.013
90 0.0075 0.003
/
KVTABT2
105 0.0055 0.0007
150 0.0045 0.0003
/
```

This example sets $K_i(p,T)$ tables for two temperature values entered using KVTEMP (see 6.8.5).

6.8.7   KVTABLIM

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets $K_i(p,T)$ tables for components. ($K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases.) For each component one should use individual keyword KVTABLIM.

The following parameters are to be specified:

1. Component number.

2. The number of columns in $K_i(p,T)$ table.

3. The number of rows in $K_i(p,T)$ table (this value can be defaulted, then the number of rows will be calculated automatically).
   The parameters 1-3 should be terminated with a slash /.

4. The minimal value of pressure for $K_i(p,T)$ (kPa) – $plow$.

5. The maximal value of pressure for $K_i(p,T)$ (kPa) – $phight$.

6. The minimal value of temperature for $K_i(p,T)$ ($°C$) – $Tlow$.

7. The maximal value of temperature for $K_i(p,T)$ ($°C$) – $Thight$.
   The parameters 4-7 should be terminated with a slash /.

After these parameters one should specify a $K_i(p,T)$ table, teminated with a slash /.

$$\begin{matrix} K(Tlow,plow) & \dots & K(Tlow,phight) \\ \dots & \dots & \dots \\ K(Thight,plow) & \dots & K(Thight,phight) \end{matrix}$$

Interpolation between table entries:

- between two $K_i(p,T)$ for two adjacent pressures: $K_i(p,T)$ varies linearly with the coefficient $1/p$;

- between two non-zero $K_i(p,T)$ for two adjacent temperatures: $ln(K_i(p,T))$ varies linearly with the coefficient $1/T$;

- between two $K_i(p,T)$ (one of the value entries is zero) for two adjacent temperatures: $K_i(p,T)$ varies linearly with the coefficient $1/T$.

Example
```
KVTABLIM
3 2 1* /
10 500 20 250 /
0.0001 0.0008
0.01 0.09
/
```

In this example the $K_i(p, T)$ table for 3-rd component has 2 rows and 2 columns.

6.8.8   KVWI

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.

If KVWI is specified then $K_i(p,T)$ - values for oil components are calculated using the following correlation formula ($K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases):

$$K_i(p,T) = \frac{p_{ci}}{p} \cdot e^{5.372697 \cdot (1+A_i) \cdot (1-T_{ci}/T)}$$

где

- T_{ci} – component critical temperature TCRIT (see 6.7.11);

- p_{ci} – component critical pressure PCRIT (see 6.7.12);

- A_i – component acentric factor ACF (see 6.7.15).

$K_i = K_i(p,T)$ -values can be entered directly using the tables KVTEMP (see 6.8.5), KVTABTn (see 6.8.6), or correlation formula KVCR (see 6.8.4).

Example
```
KWVI
```

6.8.9 HEATCR

✓	**tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the rock volumetric heat capacity value for each grid block (kJ/m3/K). The same number of values should be specified as the number of grid blocks. The data should be terminated with a slash /.

Heat capacity (H_{rock}) in each grid block is calculated as:

$$H_{rock} = C_{r0}(T - T_{ref}) + \frac{C_{r1}(T - T_{ref})^2}{2}$$

where

- T_{ref} – reference temperature;

- C_{r0} – rock volumetric heat capacity (doesn't depend on temperature), kJ/m3/K (is specified using this keyword);

- C_{r1} – temperature (T) dependence of the rock volumetric heat capacity for each grid block, kJ/m3/K/K (is specified using the keyword HEATCRT (see 6.8.10)).

```
Example
DIMENS
10 10 2
/
...
HEATCR
100*32 50*35 50*37 /
/
```

This example sets the rock volumetric heat capacity: for first 100 grid blocks – 35 (kJ/m3/K), next 50 blocks – 35 (kJ/m3/K) and the remaining 50 blocks – 37 (kJ/m3/K).

6.8.10 HEATCRT

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the temperature dependence of the rock volumetric heat capacity for each grid block (kJ/m3/K). The same number of values should be specified as the number of grid blocks. The data should be terminated with a slash /.

Heat capacity (H_{rock}) in each grid block is calculated as:

$$H_{rock} = C_{r0}(T - T_{ref}) + \frac{C_{r1}(T - T_{ref})^2}{2}$$

where

- T_{ref} – reference temperature;

- C_{r0} – rock volumetric heat capacity (doesn't depend on temperature), kJ/m3/K (is specified using the keyword HEATCR (see 6.8.9));

- C_{r1} – temperature (T) dependence of the rock volumetric heat capacity for each grid block, kJ/m3/K/K (is specified using this keyword).

```
Example
DIMENS
10 10 2
/
...
HEATCR
100*0.0004 50*0.0003 50*0.0001 /
/
```

This example sets the temperature dependence of the rock volumetric heat capacity: for first 100 grid blocks – 0.0004 (kJ/m3/K/K), next 50 blocks – 0.0003 (kJ/m3/K/K) and the remaining 50 blocks – 0.0001 (kJ/m3/K/K).

6.8.11 HEATTCR

√ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the rock heat capacity. The value of rock heat capacity is constant for every rock region (the keyword ROCKNUM (see 6.4.6) specifies for every grid block the number of rock region - transmissibility dependence on pressure table - to which it belongs).

The same number of values should be specified as the number of rock regions (the second parameter of the keyword ROCKCOMP (see 6.1.19)). The data for each region should be terminated with a slash /.

The keyword has an Eclipse compatible analogues HEATCR (see 6.8.9), HEATCRT (see 6.8.10).

The following parameters are to be specified:

1. C_{r0} the rock volumetric heat capacity (doesn't depend on temperature), kJ/m3/K;

2. C_{r1} temperature T dependence of the rock volumetric heat capacity, kJ/m3/K/K.

Heat capacity (H_{rock}) in each grid block is calculated as:

$$H_{rock} = C_{r0}(T - T_{ref}) + \frac{C_{r1}(T - T_{ref})^2}{2}$$

где T_{ref} - reference temperature.

Default:

1. the rock volumetric heat capacity – not specified,

2. temperature dependence of the rock volumetric heat capacity – 0.

```
Example
ROCKCOMP
0 2 0
/
...
HEATTCR
35 0.1 /
40 /
/
```

In this example there are 2 rock regions. First rock region: the rock volumetric heat capacity – 35, a temperature dependence of the rock volumetric heat capacity - 0.1.
Second rock region: the rock volumetric heat capacity – 40, temperature dependence of the rock volumetric heat capacity – default: 0.

6.8.12 HEATVAP

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the second A_c in the formula below (kJ/kg). The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

The enthalpy of a gaseous oil component is calculated as

$$HV_c(T) = A_c \cdot (1 - T/T_{c,crit})^{B_c} = A_c'(T_{c,crit} - T)^{B_c}, \quad A_c' = A_c/T_{c,crit}^{B_c}$$

where:
In e300 data format A_c is specified using the keyword HEATVAP (default: $0kJ/kg$), In stars data format A_c' is specified using the keyword HVR (see 8.4.21) (default: $0.25Btu/lbmol/F = 0.25 * 1.05506/0.453592 * 1.8kJ/mol/C = 1.0467kJ/mol/C$).
B_c (default: 0.38) is specified using the keyword HEATVAPE (see 6.8.13) (e300), EV (see 8.4.22) (stars).
Critical temperature of the component $T_{c,crit}$ is specified using the keyword TCRIT (see 6.7.11) (e300), TCRIT (see 8.4.12) (stars).
If $T \geq T_{c,crit}$ then $HV_c(T) = 0$.

If the component enthalpy in the liquid phase and vaporization enthalpy is specified then the gas phase enthalpy is calculated from the equation (3.34).

If the component enthalpy in the gas phase and vaporization enthalpy is specified then the oil phase enthalpy is calculated from the equation (3.34).
Default: 0.

Example
```
HEATVAP
322 118 109 /
```

This example specifies the coefficient A_c for three components.

6.8.13 HEATVAPE

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the second B_c in the formula below. The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

The enthalpy of a gaseous oil component is calculated as

$$HV_c(T) = A_c \cdot (1 - T/T_{c,crit})^{B_c} = A'_c(T_{c,crit} - T)^{B_c}, \quad A'_c = A_c/T_{c,crit}^{B_c}$$

where:
In e300 data format A_c is specified using the keyword HEATVAP (see 6.8.12) (default: $0kJ/kg$), In stars data format A'_c is specified using the keyword HVR (see 8.4.21) (default: $0.25Btu/lbmol/F = 0.25 * 1.05506/0.453592 * 1.8kJ/mol/C = 1.0467kJ/mol/C$).
B_c (default: 0.38) is specified using the keyword HEATVAPE (e300), EV (see 8.4.22) (stars).
Critical temperature of the component $T_{c,crit}$ is specified using the keyword TCRIT (see 6.7.11) (e300), TCRIT (see 8.4.12) (stars).
If $T \geq T_{c,crit}$ then $HV_c(T) = 0$.

If the component enthalpy in the liquid phase and vaporization enthalpy is specified then the gas phase enthalpy is calculated from the equation (3.34).
If the component enthalpy in the gas phase and vaporization enthalpy is specified then the oil phase enthalpy is calculated from the equation (3.34).
Default: 0.38.

Example
```
HEATVAPE
3* /
```

This example specifies default values 0.38 for the coefficient B_c for three components.

6.8.14   THCONR

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the rock thermal conductivity (kJ/m/C/day). The same number of values should be specified as the number of grid blocks. The data should be terminated with a slash /.

```
Example
DIMENS
20 20 10
/
...
THCONR
4000*48 /
/
```

In this example the rock thermal conductivity is equal to 48 for all grid blocks.

6.8.15 THCONSF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
If the keyword is enable the thermal conductivity (specified using THCONR (see 6.8.14)) is multiplied by a multiplier $1 - \alpha S_G$.
The value of α shouldn't be less than 0 and greater than 1. α is specified using this keyword. The same number of values should be specified as the number of grid blocks. The data should be terminated with a slash /.

```
Example
DIMENS
10 10 2
/
...
THCONSF
200*0.01 /
/
```

This example set the value of α equal to 0.01 for all grid blocks.

6.8.16 ROCKT

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the rock compressibility factor for each PVT region (this factor depends on the temperature).

Two parameters are to be specified in each row:

1. effective thermal expansion coefficient of the formation (c_T),

2. pressure-temperature cross-term coefficient of the formation effective porosity (c_{pT}).

Ecah row should be terminated with a slash /. The same number of rows as the number of PVT regions should be specified (TABDIMS (see 6.1.13)).

Default: none.

In stars data format **the porosity** $\phi = \phi(p,T,x,y,z)$ is

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref}))$$

where

- $\psi(x,y,z)$ – **net to gross factor** (NETPAY);

- $\phi(x,y,z)$ – porosity at the pressure p_{ref} (POR);

- c_p, c_T, c_{pT} – compressibilities (CPOR, CTPOR, CPTPOR) (default: 0); tNavigator also uses the keyword ROCKT (see 6.8.16) to specify c_T, c_{pT};

- p_{ref} – reference pressure for $\phi(x,y,z)$ (PRPOR) (default – the pressure in the first active grid block).

- T_{ref} – reference temperature TEMR (see 8.4.4).

```
Example
TABDIMS
1 2 2* 1
/
...
ROCKT
0.01  4.0E-0005
/
0.012  4.0E-0005
/
```

6.8.17 THCONT

√	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the rock thermal conductivity and the phase thermal conductivities. The value is constant in each rock region. (The keyword ROCKNUM (see 6.4.6) specifies for every grid block the number of rock region - transmissibility dependence on pressure table - to which it belongs).

The parameters of this keyword are used in the calculations of thermal conductivity of the grid block – 3.27.

The same number of values should be specified as the number of rock regions (the second parameter of the keyword ROCKCOMP (see 6.1.19)). The data for each region should be terminated with a slash /.

The keyword has an Eclipse compatible analogues THCONR (see 6.8.14), THCONSF (see 6.8.15).

The following parameters are to be specified (for each rock region):

1. rock thermal conductivity, kJ/m/C/day;

2. water thermal conductivity, kJ/m/C/day (isn't used in case if there is no water phase: two phase model oil-gas);

3. oil thermal conductivity, kJ/m/C/day (isn't used in case if there is no oil phase: two phase model water-gas);

4. gas thermal conductivity, kJ/m/C/day (isn't used in case if there is no gas phase: two phase model oil-water);

5. solid phase thermal conductivity, kJ/m/C/day (isn't used in case if there is no solid phase).

Default:

- parameters 1-4 – non specified,

- solid phase thermal conductivity is equal to the rock thermal conductivity (the first parameter of this keyword).

```
Example
ROCKCOMP
0 2 0
/
...
THCONT
160 54 10 250 /
160 54 10 /
/
```

In this example there are 2 rock regions. First region: rock thermal conductivity - 160, water thermal conductivity - 54, oil thermal conductivity - 10, gas thermal conductivity - 250.

Second region: rock thermal conductivity - 160, water thermal conductivity - 54, oil thermal conductivity - 10.

6.8.18 THCONMIX

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the method of block thermal conductivity calculation. Two methods are possible:

- SIMPLE

- COMPLEX

In stars data format **thermal conductivity of the grid block** is specified using the keyword THCONMIX (see 8.3.13) option SIMPLE

$$K_b = \phi \left(1 - \widehat{S}_S \right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

where

- k_P, $P = W, O, G, S$ – phase thermal conductivity (THCONW (see 8.3.9), THCONO (see 8.3.10), THCONG (see 8.3.11), THCONS (see 8.3.12)) (default THCONS (see 8.3.12) = THCONR (see 8.3.8));

- S_P, $P = W, O, G$ – phase saturation, $\widehat{S}_S$ – solid phase saturation;

- k_R – rock thermal conductivity (THCONR (see 8.3.8)) (default $149.6 kJ/m/day/C$);

- ϕ – porosity.

tNavigator also uses the keyword THCONT (see 6.8.17) to specify the parameters k_P, $P = W, O, G, S$ и k_R.

In stars data format isothermal part of **the block thermal conductivity** specified using the keyword THCONMIX (see 8.3.13) option COMPLEX

$$K_b' = \left(1 - \sqrt{S_W + S_O} \right) \cdot k_G \cdot F \left(\frac{k_R}{k_G} \right) + \sqrt{S_W + S_O} \cdot k_L \cdot F \left(\frac{k_R}{k_L} \right)$$

where

$$F(x) = \exp \left(\left(0.28 - 0.32876 \cdot \log \phi_f - 0.024755 \cdot \log x \right) \log x \right), \quad K_L = \frac{k_W S_W + K_O S_O}{S_W + S_O}$$

where

- ϕ_f – "mobile" porosity.

The dependence between **the block thermal conductivity** and the temperature

$$K_b = K_b' - 1.7524 \cdot 10^{-5} (T - T_{ref}) \cdot (K_b' - 119616)$$
$$\cdot \left(K_b'\right)^{-0.64} \cdot \left(K_b' \cdot \left(1.8 \cdot 10^{-3} \cdot T\right)^{\left(-3.6784 \cdot 10^{-6} K_b'\right)} + 110644.8\right)$$

where T_{ref} is given by the keyword TEMR (see 8.4.4).

In e300 data format **thermal conductivity of the grid block** is

$$K_b = (1 - \alpha S_G) \cdot k_R$$

where

- k_R – rock thermal conductivity (THCONR (see 6.8.14)) ($kJ/m/day/{}^\circ C$);

- α – is set via THCONSF (see 6.8.15), $\alpha \in [0,1]$ (default: 0);

- S_G – gas saturation.

Example
```
THCONMIX SIMPLE
```

6.8.19 STHERMX1

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the thermal expansion coefficient for each component (1/C) in the solid phase.

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

In stars component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,T}(T - T_{ref}) + c_{k,pT}(p - p_{ref})(T - T_{ref})\right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SOLID_DEN (see 8.4.13))

4. $c_{k,p}$, $c_{k,T}$, $c_{k,pT}$ – properties of component k in the solid phase (SOLID_DEN (see 8.4.13)):

   - $c_{k,p}$ – component k compressibility,
   - $c_{k,T}$ – thermal expansion coefficient for component k (for this parameter tNavigator uses this keyword STHERMX1),
   - $c_{k,pT}$ – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses this keyword STHERMX2 (see 6.8.20))

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref}))\right)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (is specified using this keyword STHERMX1)

Example
```
STHERMX1
0.0000024 0.0000033 2*
/
```

This example sets the thermal expansion coefficient for 4 components in the solid phase: 1-st component – 0.0000024, 2-nd – 0.0000033, 3-rd and 4-th components have the default value 0.

6.8.20 STHERMX2

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the coefficient of density dependence on temperature and pressure for each component in the solid phase (1/(kPa*C)).

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

Default: 0.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In stars component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,T}(T - T_{ref}) + c_{k,pT}(p - p_{ref})(T - T_{ref})\right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SOLID_DEN (see 8.4.13))

4. $c_{k,p}$, $c_{k,T}$, $c_{k,pT}$ – properties of component k in the solid phase (SOLID_DEN (see 8.4.13)):

   - $c_{k,p}$ – component k compressibility,

   - $c_{k,T}$ – thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword STHERMX1 (see 6.8.19)),

   - $c_{k,pT}$ – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses this keyword STHERMX2)

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref}))\right)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (is specified using the keyword STHERMX1 (see 6.8.19))

Example
```
STHERMX2
0.0000011 0.0000017 2*
/
```

This example sets the coefficient of density dependence on temperature and pressure for 4 components in the solid phase: 1-st component – 0.0000011, 2-nd – 0.0000017, 3-rd and 4-th components have the default value 0.

6.8.21 SDREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the density for each component in the solid phase at reference pressure and reference temperature (kg/m^3).

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF)

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (STHERMX1 (see 6.8.19))

Example
```
SDREF
35.2 48.13
/
```

This example sets density for two components in the solid phase at reference pressure and reference temperature: 1-st component has the density 35.2, 2-nd component – 48.13.

6.8.22 SPREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the reference pressure for each component in the solid phase (Bars).

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF)

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (STHERMX1 (see 6.8.19))

Example
```
SPREF
3500 3200
/
```

This example sets the reference pressure for two components: for 1-st component – 3500, for 2-nd component – 3200.

6.8.23 SCREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the component compressibility for each component in the solid phase (1/Bars).

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF (see 6.8.24))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF)

5. $c_{k,T}$ – thermal expansion coefficient for component k (STHERMX1 (see 6.8.19))

Example
```
SCREF
0.000045 0.000053
/
```

This example sets the component compressibility for two components: for 1-st component – 0.000045, for 2-nd component – 0.000053.

6.8.24　STREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the reference temperature for each component in the solid phase ($^\circ$K).

The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 component k solid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (SPREF (see 6.8.22))

2. $T_{k,ref}$ – reference temperature for component k (STREF)

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (SDREF (see 6.8.21))

4. $c_{k,p}$ – component k compressibility (SCREF (see 6.8.23))

5. $c_{k,T}$ – thermal expansion coefficient for component k (STHERMX1 (see 6.8.19))

Example
```
STREF
450 530
/
```

This example sets the reference temperature for two components: for 1-st component – 450°K, for 2-nd component – 530°K.

6.8.25 THERMEX1

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword sets the first thermal expansion coefficient for each component in the liquid phase (1/C). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

In stars component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,1,T}(T - T_{ref}) - c_{k,2,T}(T^2 - T_{ref}^2)/2 + c_{k,pT}(p - p_{ref})(T - T_{ref})\right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (MOLDEN (see 8.4.5))

4. $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – properties of component k in the liquid phase:

   - $c_{k,p}$ (CP (see 8.4.7)) – component k compressibility k,

   - $c_{k,1,T}$ (CT1 (see 8.4.8)) – the first thermal expansion coefficient for component k (for this parameter tNavigator uses this keyword THERMEX1 (see 6.8.25)),

   - $c_{k,2,T}$ (CT2 (see 8.4.9)) – the second thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword THERMEX2 (see 6.8.26)); total thermal expansion coefficient is equal to $c_{k,1,T} + T * c_{k,2,T}$,

   - $c_{k,pT}$ (CPT (see 8.4.10)) – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses the keyword THERMEX3 (see 6.8.27))

In e300 component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref}))\right)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
THERMEX1
0.0000024 0.0000033 2*
/
```

This example sets the first thermal expansion coefficient for four components in the liquid phase: for 1-st component – 0.0000024, for 2-nd component – 0.0000033, for 3-rd and 4-th components – the default value 0.

6.8.26 THERMEX2

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the second thermal expansion coefficient for each component in the liquid phase ($1/C^2$). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

In stars component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,1,T}(T - T_{ref}) - c_{k,2,T}(T^2 - T_{ref}^2)/2 + c_{k,pT}(p - p_{ref})(T - T_{ref})\right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (MOLDEN (see 8.4.5))

4. $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – properties of component k in the liquid phase:

   - $c_{k,p}$ (CP (see 8.4.7)) – component k compressibility k,
   - $c_{k,1,T}$ (CT1 (see 8.4.8)) – the first thermal expansion coefficient for component k (for this parameter tNavigator uses this keyword THERMEX1 (see 6.8.25)),
   - $c_{k,2,T}$ (CT2 (see 8.4.9)) – the second thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword THERMEX2 (see 6.8.26)); total thermal expansion coefficient is equal to $c_{k,1,T} + T * c_{k,2,T}$,
   - $c_{k,pT}$ (CPT (see 8.4.10)) – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses the keyword THERMEX3 (see 6.8.27))

In e300 component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref}))\right)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
THERMEX2
0.00000024 0.00000033 2*
/
```

This example sets the second thermal expansion coefficient for four components in the liquid phase: for 1-st component – 0.00000024, for 2-nd component – 0.00000033, for 3-rd and 4-th components – the default value 0.

6.8.27 THERMEX3

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the coefficient of density dependence on temperature and pressure for each component in the liquid phase (1/(kPa C)). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

In stars component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} \cdot \exp\left(c_{k,p}(p - p_{ref}) - c_{k,1,T}(T - T_{ref}) - c_{k,2,T}(T^2 - T_{ref}^2)/2 + c_{k,pT}(p - p_{ref})(T - T_{ref})\right)$$

where

1. p_{ref} – reference pressure (PRSR (see 8.4.3))

2. T_{ref} – reference temperature (TEMR (see 8.4.4))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (MOLDEN (see 8.4.5))

4. $c_{k,p}$, $c_{k,1,T}$, $c_{k,2,T}$, $c_{k,pT}$ – properties of component k in the liquid phase:

   - $c_{k,p}$ (CP (see 8.4.7)) – component k compressibility k,

   - $c_{k,1,T}$ (CT1 (see 8.4.8)) – the first thermal expansion coefficient for component k (for this parameter tNavigator uses this keyword THERMEX1 (see 6.8.25)),

   - $c_{k,2,T}$ (CT2 (see 8.4.9)) – the second thermal expansion coefficient for component k (for this parameter tNavigator uses the keyword THERMEX2 (see 6.8.26)); total thermal expansion coefficient is equal to $c_{k,1,T} + T * c_{k,2,T}$,

   - $c_{k,pT}$ (CPT (see 8.4.10)) – the coefficient of density dependence on temperature and pressure (for this parameter tNavigator uses the keyword THERMEX3 (see 6.8.27))

In e300 component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref}/\left((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref}))\right)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
THERMEX3
0.0000024 0.0000033 2*
/
```

This example sets the coefficient of density dependence on temperature and pressure for four components in the liquid phase: for 1-st component – 0.0000024, for 2-nd component – 0.0000033, for 3-rd and 4-th components – the default value 0.

6.8.28 PREF

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets the reference pressure for each component in the liquid phase (Bars). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF)

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
PREF
1800 1625
/
```

This example sets the reference pressure for two components in the liquid phase: for 1-st component – 1800, for 2-nd component – 1625.

6.8.29 CREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the compressibility for each component in the liquid phase (1/Bars). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
CREF
0.0000049 0.0000055
/

This example sets the compressibility for two components in the liquid phase: for 1-st component – 0.0000049, for 2-nd component – 0.0000055.

6.8.30 TREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the reference temperature for each component in the liquid phase ($^{/circ}$K). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF)

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF (see 6.8.31))

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
TREF
490 505
/
```

This example sets the reference temperature for two components in the liquid phase: for 1-st component – $490^{/circ}$K, for 2-nd component – $505^{/circ}$K.

6.8.31 DREF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the density at reference pressure and reference temperature for each component in the liquid phase (kg/m^3). The same number of values as the number of components should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Component k liquid density at pressure p and temperature T is

$$\rho_k = \rho_{k,ref} / \Big((1 - c_{k,p}(p - p_{k,ref}))(1 + c_{k,T}(T - T_{k,ref})) \Big)$$

where

1. $p_{k,ref}$ – reference pressure for component k (PREF (see 6.8.28))

2. $T_{k,ref}$ – reference temperature for component k (TREF (see 6.8.30))

3. $\rho_{k,ref}$ – density of component k at reference pressure and reference temperature (DREF)

4. $c_{k,p}$ – component k compressibility (CREF (see 6.8.29)),

5. $c_{k,T}$ – thermal expansion coefficient for component k (THERMEX1 (see 6.8.25))

Example
```
DREF
49.2 39.3 37.4
/
```

This example sets the density of three components at reference pressure and reference temperature.

6.8.32 ZFACTOR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the coefficients $Z_{k,0}$ for each component in the formula below. (The coefficients $Z_{k,1}$ are set via the keyword ZFACT1 (see 6.8.33).) The same number of values as the number of components N should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In `e300` data format **the molar** and **the gas mass density** are calculated

$$\xi_G = 1 \left/ \left(x_{w,G} \frac{MW_w}{\rho_w} + \sum_{k=2}^{n'_c} x_{k,G} \left(\frac{Z_{k,0}RT}{p} - Z_{k,1} \right) \right) \right.$$

$$\rho_G = 1 \left/ \left(\frac{x_{w,G}}{\rho_w} + \sum_{k=2}^{n'_c} \frac{x_{k,G}}{MW_k} \left(\frac{Z_{k,0}RT}{p} - Z_{k,1} \right) \right) \right.$$

The component water density (water vapor) $\rho_w(p,T)$ in the gas phase

$$\rho_w = \exp\left(\sum_{j=0}^{5} C_j T_{bK}^j \right) \frac{T_b}{T}$$

where T_b – the boiling temperature $^\circ C$, $T_{bK} = T_b + 273.15$ – the boiling temperature $^\circ K$,

$$T_b = a \cdot (p/10)^b, \quad\quad a = 180.89 \quad\quad b = 0.2350$$
$$C_0 = -93.7072 \quad\quad C_1 = 0.833941 \quad\quad C_2 = -0.003208$$
$$C_3 = 6.57652 \cdot 10^{-6} \quad\quad C_4 = -6.93747 \cdot 10^{-9} \quad\quad C_5 = 2.97203 \cdot 10^{-12}$$

The coefficients $Z_{k,0}$, $Z_{k,1}$ are specified using the keywords ZFACTOR, ZFACT1 (see 6.8.33).

In `stars` data format **the molar** and **the gas mass density** are calculated

$$\xi_G = \frac{p}{ZRT}, \quad \rho_G = \xi_G \cdot \sum_{k=1}^{n'_c} x_{k,G} \cdot MW_k$$

where Z – the root of the equation of state Redlich-Kwong with zero coefficients of pair-wise interaction. Let's specify $i = 1, \ldots, n'_c$

$$A_i = 0.4274802 \frac{p_{ri}}{T_{ri}^{2.5}}, \quad B_i = 0.08664035 \frac{p_{ri}}{T_{ri}}, \quad A_{jk} = (A_j A_k)^{0.5}, \quad p_{ri} = \frac{p}{p_{ci}}, \quad T_{ri} = \frac{T}{T_{ci}},$$

where the critical temperature T_{ci} is set using TCRIT (see 8.4.12), the creitical pressure p_{ci} is set using PCRIT (see 8.4.11). Then

$$A = \sum_{j=1}^{n'_c} \sum_{k=1}^{n'_c} x_{j,G} x_{k,G} A_{jk}, \quad B = \sum_{j=1}^{n'_c} x_{j,G} B_j$$

Z – the maximal root (>0) of the equation

$$Z^3 - Z^2 + (A - B^2 - B)Z - AB = 0$$

Z is calculated in every grid block at every time step of Newton iteration. Usually $Z \in [0.3, 1.2]$.

Example
```
ZFACTOR
5*0.96
/
```

This example sets the coefficients $Z_{k,0}$ of five components equal to 0.96.

6.8.33 ZFACT1

	tNavigator	✓	eclipse 300		MORE Roxar
	eclipse 100		CMG imex		CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the coefficients $Z_{k,1}$ for each component in the formula below. (The coefficients $Z_{k,0}$ are set via the keyword ZFACTOR (see 6.8.32).) The same number of values as the number of components N should be specified (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the molar** and **the gas mass density** are calculated

$$\xi_G = 1 \left/ \left(x_{w,G} \frac{MW_w}{\rho_w} + \sum_{k=2}^{n'_c} x_{k,G} \left(\frac{Z_{k,0}RT}{p} - Z_{k,1} \right) \right) \right.$$

$$\rho_G = 1 \left/ \left(\frac{x_{w,G}}{\rho_w} + \sum_{k=2}^{n'_c} \frac{x_{k,G}}{MW_k} \left(\frac{Z_{k,0}RT}{p} - Z_{k,1} \right) \right) \right.$$

The component water density (water vapor) $\rho_w(p,T)$ in the gas phase

$$\rho_w = \exp\left(\sum_{j=0}^{5} C_j T_{bK}^j \right) \frac{T_b}{T}$$

where T_b – the boiling temperature $^\circ C$, $T_{bK} = T_b + 273.15$ – the boiling temperature $^\circ K$,

$T_b = a \cdot (p/10)^b$,	$a = 180.89$	$b = 0.2350$
$C_0 = -93.7072$	$C_1 = 0.833941$	$C_2 = -0.003208$
$C_3 = 6.57652 \cdot 10^{-6}$	$C_4 = -6.93747 \cdot 10^{-9}$	$C_5 = 2.97203 \cdot 10^{-12}$

The coefficients $Z_{k,0}$, $Z_{k,1}$ are specified using the keywords ZFACTOR (see 6.8.32), ZFACT1.

In stars data format **the molar** and **the gas mass density** are calculated

$$\xi_G = \frac{p}{ZRT}, \quad \rho_G = \xi_G \cdot \sum_{k=1}^{n'_c} x_{k,G} \cdot MW_k$$

where Z – the root of the equation of state Redlich-Kwong with zero coefficients of pair-wise interaction. Let's specify $i = 1, \ldots, n'_c$

$$A_i = 0.4274802 \frac{p_{ri}}{T_{ri}^{2.5}}, \quad B_i = 0.08664035 \frac{p_{ri}}{T_{ri}}, \quad A_{jk} = (A_j A_k)^{0.5}, \quad p_{ri} = \frac{p}{p_{ci}}, \quad T_{ri} = \frac{T}{T_{ci}},$$

where the critical temperature T_{ci} is set using TCRIT (see 8.4.12), the creitical pressure p_{ci} is set using PCRIT (see 8.4.11). Then

$$A = \sum_{j=1}^{n'_c} \sum_{k=1}^{n'_c} x_{j,G} x_{k,G} A_{jk}, \quad B = \sum_{j=1}^{n'_c} x_{j,G} B_j$$

Z – the maximal root (> 0) of the equation

$$Z^3 - Z^2 + (A - B^2 - B)Z - AB = 0$$

Z is calculated in every grid block at every time step of Newton iteration. Usually $Z \in [0.3, 1.2]$.

Example
```
ZFACT1
5*0.15
/
```

This example sets the coefficients $Z_{k,1}$ of five components equal to 0.15.

6.8.34 VISCREF

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets reference pressure and dissolved gas concentration for each PVT region.
In `e300` data format the keyword should be present if oil or water viscosity depends on pressure.

For each PVT region the following parameters should be entered:

1. reference pressure (*barsa*);

2. reference dissolved gas consectration (sm^3/sm^3) (DISGAS (see 6.1.31));

3. IGNORED, this is an Eclipse compatibility field

The data for each PVT region should be terminated with a slash /.

```
Example
VISCREF
2500 1.5/
3200 1.3/
```

In this example reference pressure and dissolved gas concentration are specified for 2 PVT regions.

6.8.35 WATVISCT

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of water viscosity as a function of temperature for each PVT region (The number of PVT regions is equal to the second parameter of TABDIMS (see 6.1.13)).

For each PVT region the table should be terminated with a slash /. Each row of the table consists of two parameters:

1. temperature;

2. water viscosity at this temperature $\mu_W(T)$.

Viscosity dependence on pressure can be specified:

$$\mu_W(p,T) = \mu_W(T)\frac{\mu'_W(p)}{\mu'_W(p_{ref})}$$

- $\mu_W(T)$ – water viscosity (depends on temperature);

- $\mu'_W(p)$ – water viscosity as a function ot pressure (specified using PVTW (see 6.5.4));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

Example
```
WATVISCT
10 1
60 0.8
130 0.45
/
```

This example sets the table of water viscosity as a function of temperature for one PVT region.

6.8.36 OILVISCT

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of oil component viscosity as a function of temperature for each PVT region (The number of PVT regions is equal to the second parameter of TABDIMS (see 6.1.13)).

For each PVT region the table should be terminated with a slash /. Each row of the table consists of parameters:

- temperature;

- N values of oil component viscosities for this temperature $\mu_{k,O}(T)$ (N – the number of components COMPS (see 6.7.3)).

Oil viscosity is calculated using the formula

$$\mu_O(T) = \prod_{k=2}^{n'_c} (\mu_{k,O})^{f_k(x_{k,o})}$$

where **oil component viscosity** $\mu_{k,O}(T)$ can be specified as a function of temperature using the tables of this keyword OILVISCT (in e300 data format), VISCTABLE (see 8.4.38) (stars), or using correlation (OILVISCC (see 6.8.37)).
$f_k(x)$ (default: $f_k(x) = x$) are specified using OILVINDX (see 6.8.38) (in e300 data format) or VSMIXCOMP (see 8.4.39), VSMIXENDP (see 8.4.40), VSMIXFUNC (see 8.4.41) (stars).

In e300 data format pressure dependence can be set:

$$\mu_O(p,T) = \mu_O(T) \frac{\mu'_O(p)}{\mu'_O(p_{ref})}$$

where

- $\mu_O(T)$ – oil viscosity (depends on temperature);

- $\mu'_O(p)$ – oil viscosity as a function of pressure (specified using PVCO (see 6.5.5));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

```
Example
OILVISCT
60 2500 4700
100 950 1200
130 115 161
250 14 17
370 3.3 3.9
500 1.8 2.1
/
```

This example specifies the table of oil component viscosity as a function of temperature for one PVT region.

6.8.37 OILVISCC

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of coefficients for correlation formula (oil viscosity as a function of temperature).

Oil viscosity is calculated using the formula

$$\mu_O(T) = \prod_{k=2}^{n'_c}(\mu_{k,O})^{f_k(x_{k,o})}$$

where **oil component viscosity** $\mu_{k,O}(T)$ can be specified as a function of temperature using the tables of the keyword OILVISCT (see 6.8.36) (in e300 data format), VISCTABLE (see 8.4.38) (stars), or using correlation (this keyword OILVISCC).
$f_k(x)$ (default: $f_k(x) = x$) are specified using OILVINDX (see 6.8.38) (in e300 data format) or VSMIXCOMP (see 8.4.39), VSMIXENDP (see 8.4.40), VSMIXFUNC (see 8.4.41) (stars).

In e300 data format the following correlations can be used to calculate $\mu_{k,O}$ (specified using this keyword OILVISCC):

Name	Formula	Name	Formula
ASTM	$\log_{10}(\mu_{k,O} + A_k) = B_k T^{C_k}$	Vogel	$\log_{10}(\mu_{k,O}) = A_k + B_k/(T + C_k)$
Andrade	$\log_{10}(\mu_{k,O}) = A_k + B_k/T$	logarithmic	$\log_{10}(\mu_{k,O}) = A_k + B_k\log_{10}(T)$

In stars data format Andrade correlation is used with parameters $A_k = \log_{10}A'_k$, $B_k = B'_k\log_{10}e$, where A'_k, B'_k are specified using keywords AVISC (see 8.4.36), BVISC (see 8.4.37).

The coefficients of correlation formula should be entered as several rows. First row consists of the name of correlation formula. Other rows contain the coefficients A, B, C. The data should be terminated with a slash /.

ASTM:

1. ASTM FORMULA

2. coefficient A (N (COMPS (see 6.7.3)) values: each value for one component; $A < 1$

3. coefficient B (N (COMPS (see 6.7.3)) values: each value for one component; $B > 0$

4. coefficient C (N (COMPS (see 6.7.3)) values: each value for one component; $C > 0$

Andrade:

1. ANDRADE FORMULA

2. coefficient A (N (COMPS (see 6.7.3)) values: each value for one component);

3. coefficient B (N (COMPS (see 6.7.3)) values: each value for one component);

Vogel:

1. VOGEL FORMULA

2. coefficient A (N (COMPS (see 6.7.3)) values: each value for one component);

3. coefficient B (N (COMPS (see 6.7.3)) values: each value for one component);

4. coefficient C (N (COMPS (see 6.7.3)) values: each value for one component);

Logarithmic:

1. LOG FORMULA

2. coefficient A (N (COMPS (see 6.7.3)) values: each value for one component);

3. coefficient B (N (COMPS (see 6.7.3)) values: each value for one component);

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)).

```
Example
OILVISCC
ASTM FORMULA
0.4 0.5 0.6
12 18 20
11 21 24
/
```

This example sets the coefficients for ASTM correlation formula for one EoS region.

6.8.38   OILVINDX

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword sets the table of functions $f_k(x)$ (indices in oil viscosity formula below) for each PVT region (The number of PVT regions is equal to the second parameter of TABDIMS (see 6.1.13)).

For each PVT region the table should be terminated with a slash /. Each row of the table consists of parameters:

- mole fraction value x (the number between 0 and 1);

- N values of $f_k(x)$ (N – the number of components COMPS (see 6.7.3)).

$f_k(0) = 0, f_k(1) = 1$.

Oil viscosity is calculated using the formula

$$\mu_O(T) = \prod_{k=2}^{n'_c} (\mu_{k,O})^{f_k(x_{k,o})}$$

where **oil component viscosity** $\mu_{k,O}(T)$ can be specified as a function of temperature using the tables of the keyword OILVISCT (see 6.8.36) (in e300 data format), VISCTABLE (see 8.4.38) (stars), or using correlation (OILVISCC (see 6.8.37)).

$f_k(x)$ (default: $f_k(x) = x$) are specified using this keyword OILVINDX (in e300 data format) or VSMIXCOMP (see 8.4.39), VSMIXENDP (see 8.4.40), VSMIXFUNC (see 8.4.41) (stars). tNavigator also uses the keyword OILVINDT (see 6.8.39).

In e300 data format pressure dependence can be set:

$$\mu_O(p, T) = \mu_O(T) \frac{\mu'_O(p)}{\mu'_O(p_{ref})}$$

where

- $\mu_O(T)$ – oil viscosity (depends on temperature);

- $\mu'_O(p)$ – oil viscosity as a function of pressure (specified using PVCO (see 6.5.5));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

Example
```
OILVINDX
0 0 0
0.1 0.01 0.02
0.2 0.03 0.04
0.3 0.07 0.08
0.4 0.09 0.12
0.5 0.18 0.22
0.6 0.28 0.39
0.7 0.50 0.51
0.8 0.73 0.67
0.9 0.82 0.89
1 1 1
/
```

This example specifies the table of functions $f_k(x)$ for one PVT region.

6.8.39 OILVINDT

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of functions $f_k(x)$ (indices in oil viscosity formula below) for each PVT region (The number of PVT regions is equal to the second parameter of TABDIMS (see 6.1.13)).

For each PVT region the following table should be specified.

$$
\begin{array}{ll}
x_{1,1} & f_1(x_{1,1}) \\
\ldots & \ldots \\
x_{1,11} & f_1(x_{1,11}) \\
/ & \\
x_{2,1} & f_2(x_{2,1}) \\
\ldots & \ldots \\
x_{2,11} & f_2(x_{2,11}) \\
/ & \\
\ldots & \ldots \\
/ & \\
x_{N,1} & f_N(x_{N,1}) \\
\ldots & \ldots \\
x_{N,11} & f_N(x_{N,11}) \\
/ &
\end{array}
$$

This table consists of the following parameters:

- mole fraction value $x_{i,j}$ (the number between 0 and 1), $0 \le x_{i,1} \le x_{i,2} \le \ldots \le x_{i,11} \le 1$;

- values of $f_k(x)$, $1 \le k \le N$ (N – the number of components COMPS (see 6.7.3)).

$f_k(0) = 0, f_k(1) = 1$.

If the are more than one PVT region, the tables for all regions should be specified one after another in this keyword.

Oil viscosity is calculated using the formula

$$
\mu_O(T) = \prod_{k=2}^{n'_c} (\mu_{k,O})^{f_k(x_{k,o})}
$$

where **oil component viscosity** $\mu_{k,O}(T)$ can be specified as a function of temperature using the tables of the keyword OILVISCT (see 6.8.36) (in e300 data format), VISCTABLE (see 8.4.38) (stars), or using correlation (OILVISCC (see 6.8.37)).

$f_k(x)$ (default: $f_k(x) = x$) are specified using the keyword OILVINDX (see 6.8.38) (in e300 data format) or VSMIXCOMP (see 8.4.39), VSMIXENDP (see 8.4.40), VSMIXFUNC (see 8.4.41) (stars).

tNavigator also uses the keyword OILVINDT (see 6.8.39).

In e300 data format pressure dependence can be set:

$$\mu_O(p, T) = \mu_O(T) \frac{\mu'_O(p)}{\mu'_O(p_{ref})}$$

where

- $\mu_O(T)$ – oil viscosity (depends on temperature);

- $\mu'_O(p)$ – oil viscosity as a function of pressure (specified using PVCO (see 6.5.5));

- p_{ref} – reference pressure (specified using VISCREF (see 6.8.34)).

Example
```
COMPS
2 /
...
OILVINDT
0 0 0
0.1 0.01
0.2 0.03
0.3 0.07
0.4 0.09
0.5 0.18
0.6 0.28
0.7 0.50
0.8 0.73
0.9 0.82
1 1 1
/
0 0 0
0.1 0.02
0.2 0.04
0.3 0.09
0.4 0.19
0.5 0.32
0.6 0.44
0.7 0.60
0.8 0.79
0.9 0.89
1 1 1
/
```

This example specifies the table of functions $f_k(x)$ for two components and one PVT region.

6.8.40 GASVISCT

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of gas component viscosity as a function of temperature for each PVT region (The number of PVT regions is equal to the second parameter of TABDIMS (see 6.1.13)).

For each PVT region the table should be terminated with a slash $/$. Each row of the table consists of parameters:

- temperature;

- N values of gas component viscosities for this temperature $\mu_{k,G}(T)$ (N – the number of components COMPS (see 6.7.3)).

In e300 data format **gas viscosity** is calculated

$$\mu_G(p,T) = \sum_{k=1}^{n'_c} x_{k,G}\mu_{k,G}$$

component water viscosity (water vapor) $\mu_{1,G}(p,T)$ in the gas phase

$$\mu_{1,G}(p,T) = A_g + B_g T_C + C_g (p/10)^{D_g}, \quad T_C = T - 273.15$$
$$A_g = 4.9402 \cdot 10^{-3}, \quad B_g = 5.0956 \cdot 10^{-5}, \quad C_g = 2.9223 \cdot 10^{-6}, \quad D_g = 2.5077$$

hydrocarbon component viscosity $\mu_{k,G}(T)$ in the gas phase can be specified as a function of temperature using the tables of this keyword GASVISCT, or using the correlation formula with the coefficients GASVISCF (see 6.8.41):

$$\mu_{k,G}(T) = A_k \cdot T^{B_k}$$

In stars data format **gas viscosity** is calculated as

$$\mu_G(T) = \left(\sum_{k=1}^{n'_c} \mu_{k,G} \cdot x_{k,G} \sqrt{MW_k} \right) \Bigg/ \left(\sum_{k=1}^{n'_c} x_{k,G} \sqrt{MW_k} \right)$$

where MW_k – molecular weight of the component k (specified using CMM (see 8.4.44)), component viscosities are set via correlation (3.32) with the coefficients AVG (see 8.4.42), BVG (see 8.4.43). If gas viscosity isn't specified by user then in stars data format it is calculated as

$$\mu_G(T) = 0.0136 + 3.8 \cdot 10^{-5} \cdot T_C, \quad T_C = T - 273.15$$

Example
```
GASVISCT
60 0.0122 0.017
100 0.0132 0.027
130 0.0150 0.027
250 0.0191 0.030
370 0.0210 0.033
500 0.0310 0.039
/
```

This example specifies the table of gas component viscosity as a function of temperature for one PVT region.

6.8.41 GASVISCF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the table of coefficients for correlation formula (gas component viscosity as a function of temperature).

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

The rows of table for one EoS region contains the following parameters:

- coefficient A (N (COMPS (see 6.7.3)) values: each value for one component);

- coefficient B (N (COMPS (see 6.7.3)) values: each value for one component);

In e300 data format **gas viscosity** is calculated

$$\mu_G(p,T) = \sum_{k=1}^{n'_c} x_{k,G}\mu_{k,G}$$

component water viscosity (water vapor) $\mu_{1,G}(p,T)$ in the gas phase

$$\mu_{1,G}(p,T) = A_g + B_g T_C + C_g (p/10)^{D_g}, \quad T_C = T - 273.15$$
$$A_g = 4.9402 \cdot 10^{-3}, \quad B_g = 5.0956 \cdot 10^{-5}, \quad C_g = 2.9223 \cdot 10^{-6}, \quad D_g = 2.5077$$

hydrocarbon component viscosity $\mu_{k,G}(T)$ in the gas phase can be specified as a function of temperature using the tables of this keyword GASVISCT, or using the correlation formula with the coefficients GASVISCF (see 6.8.41):

$$\mu_{k,G}(T) = A_k \cdot T^{B_k}$$

In stars data format **gas viscosity** is calculated as

$$\mu_G(T) = \left(\sum_{k=1}^{n'_c} \mu_{k,G} \cdot x_{k,G} \sqrt{MW_k} \right) \Big/ \left(\sum_{k=1}^{n'_c} x_{k,G} \sqrt{MW_k} \right)$$

where MW_k – molecular weight of the component k (specified using CMM (see 8.4.44)), component viscosities are set via correlation with the coefficients AVG (see 8.4.42), BVG (see 8.4.43). If gas viscosity isn't specified by user then in stars data format it is calculated as

$$\mu_G(T) = 0.0136 + 3.8 \cdot 10^{-5} \cdot T_C, \quad T_C = T - 273.15$$

Example
```
GASVISCF
0.00022 0.00017
0.8 0.7
/
```

This example sets the coefficients for correlation formula for one EoS region.

6.8.42   REACRATE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the rate of chemical reactions (the rate is constant for each reaction). The same number of values as the number of chemical reactions n_r (REACTION (see 6.1.26)) should be specified. The data should be terminated with a slash /.

Default: 0.

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT (see 6.8.43) in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD (see 6.8.44), REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

Reaction rate of the reaction number r, $r = 1, \ldots, n_r$ ($kg - mol/day$) for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c} (c'_{ri})^{n_{ri}}$$

where $R = 8.3143 \dfrac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$c_{ri} = \begin{cases} \left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\ \widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\ N_i & \text{if the reactant } i \text{ is in all phases} \end{cases}$$

$$c'_{ri} = \begin{cases} \phi \cdot c_{ri} & \text{if it isn't gas and not } \textsf{REACPHA (see 6.8.50) with GPP (there is no O2PP (see 8.4.33))} \\ p' \cdot x_{i,G} & \text{else (for gas)} \end{cases}$$

where temperature, pressure and order are:

$$T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases} \quad p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases} \quad n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases}$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

Example
```
REACTION
4 /
...
REACRATE
0.0000042 0.0000021 0.0000035
/
```

This example sets the reaction rate for three chemical reactions, the fourth reaction has default zero rate. 4 reactions are specified using the keyword REACTION (see 6.1.26).

6.8.43   REACACT

✓	**tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the activation energy in chemical reaction rates (kJ/kg-M); the activation energy is constant for each reaction. The same number of values as the number of chemical reactions n_r (REACTION (see 6.1.26)) should be specified. The data should be terminated with a slash /.

Default: 0.

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE (see 6.8.42) in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD (see 6.8.44), REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

Reaction rate of the reaction number r, $r = 1, \ldots, n_r$ ($kg - mol/day$) for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c} (c'_{ri})^{n_{ri}}$$

where $R = 8.3143 \dfrac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$c_{ri} = \begin{cases} \left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\ \widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\ N_i & \text{if the reactant } i \text{ is in all phases} \end{cases}$$

$$c'_{ri} = \begin{cases} \phi \cdot c_{ri} & \text{if it isn't gas and not } \mathsf{REACPHA} \text{ (see 6.8.50) with GPP (there is no } \mathsf{O2PP} \text{ (see 8.4.33))} \\ p' \cdot x_{i,G} & \text{else (for gas)} \end{cases}$$

where temperature, pressure and order are:

$$T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases} \quad p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases} \quad n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases}$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

Example
```
REACTION
4 /
...
REACACT
18200 17000 12500
/
```

This example sets the activation energy in reaction rate for three chemical reactions, the fourth reaction has default zero activation energy. 4 reactions are specified using the keyword REACTION (see 6.1.26).

6.8.44 REACCORD

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the order of component terms in chemical reaction rates (this parameter can possess different values for the same reactant and different reactions).

The same number of rows as the number of chemical reactions n_r (REACTION (see 6.1.26)) should be specified. Each row should be terminated with a slash /. Each row contains the same number of values as the number of components – N (COMPS (see 6.7.3)).

If this order is 0, then the rate of this reaction is independent from the concentraion of this reactant. If this order is positive number - this reactant is catalyst (1 – there is linear dependence between the reaction rate and the concentration, 2 – quadratic dependence, etc.). If this order is negative number - this reactant is inhibitor (-1 – inversely proportional dependence, etc.).

Default: 0.

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE (see 6.8.42) in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT (see 6.8.43) in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD, REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

Reaction rate of the reaction number r, $r = 1, \ldots, n_r$ ($kg - mol/day$) for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c} (c'_{ri})^{n_{ri}}$$

where $R = 8.3143 \frac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$c_{ri} = \begin{cases} \left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\ \widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\ N_i & \text{if the reactant } i \text{ is in all phases} \end{cases}$$

$$c'_{ri} = \begin{cases} \phi \cdot c_{ri} & \text{if it isn't gas and not REACPHA (see 6.8.50) with GPP (there is no O2PP (see 8.4.33))} \\ p' \cdot x_{i,G} & \text{else (for gas)} \end{cases}$$

where temperature, pressure and order are:

$$T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases} \quad p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases} \quad n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases}$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

```
Example
REACTION
2 /
COMPS
6 /
...
REACCORD
0 0 1 1 0 0 /
1 0 0 0 -1 0 /
```

In this example two chemical reactions are specified using the keyword REACTION (see 6.1.26) and 6 components – using the keyword COMPS (see 6.7.3). The rate of first reaction depends (linearly) of the concentration of 3-rd and 4-th components. The rate of second reaction depends (linearly) of the concentration of 1-st component and depends (inversely) of the concentration of 5-th component.

6.8.45 REACLIMS

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the maximum and minimum temperature and pressure for each chemical reaction, which is used in the reaction rate calculations.

The same number of lines as the number of chemical reactions n_r (REACTION (see 6.1.26)) should be specified. Each line should be terminated with a slash /.

Default: are not specified.

One line consists of the following parameters:

1. minimum pressure p_l (*barsa*). If the pressure in the grid block is below this minimum value p_l, then p_l is used in the reaction rate calculations (according the formula below);

2. maximum pressure p_u (*barsa*). If the pressure in the grid block is above this maximum value p_u, then p_u is used in the reaction rate calculations (according the formula below);

3. minimum temperature T_l ($^\circ$C). If the temperature in the grid block is below this minimum value T_l, then T_l is used in the reaction rate calculations (according the formula below);

4. maximum temperature T_u ($^\circ$C). If the temperature in the grid block is above this maximum value T_u, then T_u is used in the reaction rate calculations (according the formula below).

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE (see 6.8.42) in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT (see 6.8.43) in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD (see 6.8.44), REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

Reaction rate of the reaction number r, $r = 1,\ldots,n_r$ $(kg-mol/day)$ for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c}(c'_{ri})^{n_{ri}}$$

where $R = 8.3143\,\dfrac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$c_{ri} = \begin{cases} \left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\ \left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\ \widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\ N_i & \text{if the reactant } i \text{ is in all phases} \end{cases}$$

$$c'_{ri} = \begin{cases} \phi \cdot c_{ri} & \text{if it isn't gas and not REACPHA (see 6.8.50) with GPP (there is no O2PP (see 8.4.33))} \\ p' \cdot x_{i,G} & \text{else (for gas)} \end{cases}$$

where temperature, pressure and order are:

$$T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases} \qquad p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases} \qquad n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases}$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

Example
```
REACTION
2 /
...
REACLIMS
20 75 120 320 /
30 80 100 220 /
```

In this example two chemical reactions are specified using the keyword REACTION (see 6.1.26). The maximum and minimum temperature and pressure are specified.

6.8.46 REACCONC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the critical value of reactant concentration (this (kPa, if the pressure is used or gmol/m^3, if the concentration is used). If the concentration of reactant is below this value, then the reaction rate depends linearly of this concentration (according the formula below).

The same number of rows as the number of chemical reactions n_r (REACTION (see 6.1.26)) should be specified. Each row should be terminated with a slash /. Each row contains the same number of values as the number of components – N (COMPS (see 6.7.3)).

Default: 0.

Let's consider n_r chemical reactions. For each r, $r = 1, \ldots, n_r$ there are:

- $S_{Rr} = (S_{Rri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for reactants** of the reaction number r, are specified using STOREAC (see 6.8.49) in e300; STOREAC (see 8.4.23) in stars;

- $S_{Pr} = (S_{Pri})_{i=1,\ldots,n_c}$ – **stoichiometric coefficients for products** of the reaction number r, are specified using STOPROD (see 6.8.48) in e300; STOPROD (see 8.4.24) in stars;

- A_r – **reaction rate** of the reaction number r, are specified using REACRATE (see 6.8.42) in e300; FREQFAC (see 8.4.25) in stars;

- E_r – **activation energy in chemical reaction rates** of the reaction number r, are specified using REACACT (see 6.8.43) in e300; EACT (see 8.4.26) in stars;

- H_r – **reaction enthalpy** of the reaction number r, are specified using REACENTH (see 6.8.52) in e300; RENTH (see 8.4.27) in stars;

- $N_r = (n_{ri})_{i=1,\ldots,n_c}$ – **order of component terms**, in chemical reaction r rate, (for non reactants can be > 0 (for catalyst), < 0 (for inhibitor)), are specified using REACCORD (see 6.8.44), REACSORD (see 6.8.51) in e300; RORDER (see 8.4.28) in stars.

Reaction rate of the reaction number r, $r = 1, \ldots, n_r$ ($kg - mol/day$) for the volume V_b is

$$R_r = V_b \cdot A_r \cdot e^{-E_r/(RT')} \cdot \prod_{i=1}^{n_c} (c'_{ri})^{n_{ri}}$$

where $R = 8.3143 \dfrac{kJ}{K \cdot kg - mol}$, component concentration (since (3.4)):

$$
c_{ri} = \begin{cases}
\left(1 - \widehat{S}_S\right) \cdot \xi_O \cdot S_O x_{iO} & \text{if the reactant } i \text{ is in the oil phase} \\
\left(1 - \widehat{S}_S\right) \cdot \xi_G \cdot S_G x_{iG} & \text{if the reactant } i \text{ is in the gas phase} \\
\left(1 - \widehat{S}_S\right) \cdot \xi_W \cdot S_W x_{iW} & \text{if the reactant } i \text{ is in the water phase} \\
\widehat{S}_S \cdot \xi_S \cdot x_{iS} = N_i & \text{if the reactant } i \text{ is in the solid phase} \\
N_i & \text{if the reactant } i \text{ is in all phases}
\end{cases}
$$

$$
c'_{ri} = \begin{cases}
\phi \cdot c_{ri} & \text{if it isn't gas and not REACPHA (see 6.8.50) with GPP (there is no O2PP (see 8.4.33))} \\
p' \cdot x_{i,G} & \text{else (for gas)}
\end{cases}
$$

where temperature, pressure and order are:

$$
T' = \begin{cases} T_u & \text{if } T > T_u \\ T_l & \text{if } T < T_l \\ T & \text{else} \end{cases}
\qquad
p' = \begin{cases} p_u & \text{if } p > p_u \\ p_l & \text{if } p < p_l \\ p & \text{else} \end{cases}
\qquad
n'_{ri} = \begin{cases} n_{ri} & \text{if } c'_{ri} > C_{ri} \\ 1 & \text{else} \end{cases}
$$

where

- T_u is specified using RTEMUPR (see 8.4.30) (REACLIMS (see 6.8.45) in e300 data format),

- T_l – RTEMLOWR (see 8.4.31) (REACLIMS (see 6.8.45) in e300 data format),

- C_{ri} – RXCRITCON (see 8.4.32) (there is no analogue in e300 ($n'_{ri} = n_{ri}$); tNavigator uses the keyword C_{ri} = REACCONC (see 6.8.46)),

- p_u, p_l is specified using REACLIMS (see 6.8.45) (there is no analogue in stars ($p' = p$)).

O2PP (see 8.4.33) is used for components in gas phase and is the default value for oxygen.

```
Example
REACTION
2 /
COMPS
6 /
...
REACCONC
0.00025 /
0.00011 0.00037 /
```

In this example two chemical reactions are specified using the keyword REACTION (see 6.1.26) and 6 components – using the keyword COMPS (see 6.7.3). For first reaction the critical value of concentration is specified for first reactant; for second reaction – for first and second reactants. Other critical values have default zero values.

6.8.47 REACPORD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the order of pore volume in chemical reaction rates – $n_{r,p}$ (formula 3.61). The same values should be specified as the number of chemical reactions. The data should be terminated with a slash /.

Chemical reaction rate could be independent of pore volume ($n_{r,p} = 0$).

```
Example
REACTION
3 /
REACPORD
0 0 1 /
```

In this example three chemical reactions are specified using the keyword REACTION (see 6.1.26). Chemical reaction rate of first and second reactions are independent of pore volume ($n_{r,p} = 0$). Chemical reaction rate of third reaction depends of pore volume linearly ($n_{r,p} = 1$).

6.8.48 STOPROD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the stoichiometric coefficients for all products in each chemical reaction as a line of numbers. The same number of lines as the number of chemical reactions (n_r) should be entered (REACTION (see 6.1.26)). Each line should be terminated with a slash /.

Each line should contain N values (number of components entered using COMPS (see 6.7.3)). If the component isn't the product of the reaction one should enter zero or default value.

Stoichiometric coefficients are specified using this keyword and STOREAC (see 6.8.49).
Default: 0.

For the chemical reaction
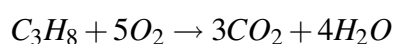
$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$

there is an example:

```
Example
REACTION
1 /
COMPS
5 /
...
STOPROD
0 0 0 3 4 /

STOREAC
0 1 5 0 0 /
```

This example sets 1 chemical reaction using the keyword REACTION (see 6.1.26) and 5 components using the keyword COMPS (see 6.7.3). The products of this reaction are: the 4-th component (with the coefficient 3) and the 5-th component (with the coefficient 4). 1-st component isn't present in this reaction.

6.8.49   STOREAC

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the stoichiometric coefficients for all reactants in each chemical reaction as a line of numbers. The same number of lines as the number of chemical reactions (n_r) should be entered (REACTION (see 6.1.26)). Each line should be terminated with a slash /.

Each line should contain N values (number of components entered using COMPS (see 6.7.3)). If the component isn't the reactant of the reaction one should enter zero or default value.

Stoichiometric coefficients are specified using this keyword and STOPROD (see 6.8.48). **Default:** 0.

For the chemical reaction
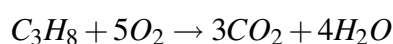
$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$

there is an example:

```
Example
REACTION
1 /
COMPS
5 /
...
STOPROD
0 0 0 3 4 /

STOREAC
0 1 5 0 0 /
```

This example sets 1 chemical reaction using the keyword REACTION (see 6.1.26) and 5 components using the keyword COMPS (see 6.7.3). The reactants of this reaction are: the 2-nd component (with the coefficient 1) and the 3-rd component (with the coefficient 5). 1-st component isn't present in this reaction.

6.8.50   REACPHA

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the component phase in chemical reaction. Each line of phases corresponds to one chemical reaction. The same number of lines as the number of chemical reactions (n_r) should be entered (REACTION (see 6.1.26)). Each line should be terminated with a slash /. Each line should contain N values (number of components entered using COMPS (see 6.7.3)). If the component isn't the reactant of the reaction one can enter default value - ALL (component phase is not used in the calculations if the component isn't reactant).

A phase can be one of the following values:

- ALL – if all phases of component react,

- OIL – oil phase reacts,

- GAS – gas phase reacts,

- GPP – gas phase reacts and the reaction rate depends on gas gas partial pressure,

- WAT – water in eater phase reacts,

- NONE – the reaction rate doesn't depend on the component.

If the component can react in different phases and the reaction rate depends on the phase, then two reactions should be specified (with the same stoichiometric coefficients – STOREAC (see 6.8.49) and different REACPHA).

Default: ALL. If the component has zero order in REACCORD the component phase will be ignored.

For the chemical reaction

$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$
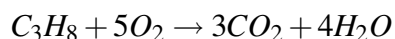
there is an example:

Example
```
REACTION
1 /
COMPS
5 /
...
STOPROD
0 0 0 3 4 /

STOREAC
0 1 5 0 0 /

REACPHA
1* OIL GAS 2* /
```

This example sets 1 chemical reaction using the keyword REACTION (see 6.1.26) and 5 components using the keyword COMPS (see 6.7.3). The reactants of this reaction are: the 2-nd component (with the coefficient 1) and the 3-rd component (with the coefficient 5). 1-st component isn't present in this reaction. The 2-nd component is in the oil phase, the 3-rd – gas phase. Other phase components are defaulted, because the reaction rate doesn't depend on them.

6.8.51 REACSORD

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the order of dependence between solid saturation and chemical reaction rate. The same number of values as the number of chemical reactions (n_r) should be entered (REACTION (see 6.1.26)).

The chemical reaction rate is proportionate to the multiplier:

$$(1 - \hat{S}_S)^{n_s}$$

where n_s – order, $\hat{S}_S$ – solid saturation.

In there is a solid phase in the model then the equatin 2.4 is replaced by the equation

$$\hat{S}_O + \hat{S}_G + \hat{S}_W + \hat{S}_S = 1$$

Oil, gas and water saturation are:

$$S_O = \frac{\hat{S}_O}{1 - \hat{S}_S}, \quad S_G = \frac{\hat{S}_G}{1 - \hat{S}_S}, \quad S_W = \frac{\hat{S}_W}{1 - \hat{S}_S}$$

Example
```
REACTION
3 /
...
REACSORD
0 1 0 /
```

6.8.52   REACENTH

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies the enthalpy of each chemical reaction (kJ/kg-mol). It can only be used if THERMAL (see 6.1.25) is present. The same number of values as the number of chemical reactions (n_r) should be entered (REACTION (see 6.1.26)). The data should be terminated with a slash /.

Default: 0.

```
Example
REACTION
2 /
...
REACENT
17000 15500 /
```

This example sets 2 chemical reactions using the keyword REACTION (see 6.1.26). Enthalpy of first reaction is 17000 kJ/kg-mol, of second reaction – 15500 kJ/kg-mol.

6.8.53 SPECHA

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the first coefficient $CP_{1,c}$ in the formula below ($kJ/kg/°C$). The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

where the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHA, SPECHB (see 6.8.54) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).
In stars data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients $CP_{i,c}$, $i = 1, \ldots, 4$ are specified using the keywords CPL1 / CPL2 / CPL3 / CPL4 (see 8.4.18). tNavigator also uses the keywords $CP_{3,c}$ = SPECHC (see 6.8.55), $CP_{4,c}$ = SPECHD (see 6.8.56). Default values: $CP_{1,c} = 0.5Btu/lbmol/F = 0.5 * 1.05506/0.453592 * 1.8kJ/mol/C = 2.0934kJ/mol/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).
Then the liquid phase enthalpy is calculated according to (3.35).

Default: 0.

Example
```
SPECHA
0.66 0.52 /
```

This example specifies the first coefficient of the formula of component liquid enthalpy for two components.

6.8.54 SPECHB

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the second coefficient $CP_{2,c}$ in the formula below ($kJ/kg/°C/°C$). The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In `e300` data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

where the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHA, SPECHB (see 6.8.54) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).
In `stars` data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients $CP_{i,c}$, $i = 1,\ldots,4$ are specified using the keywords CPL1 / CPL2 / CPL3 / CPL4 (see 8.4.18). tNavigator also uses the keywords $CP_{3,c}$ = SPECHC (see 6.8.55), $CP_{4,c}$ = SPECHD (see 6.8.56). Default values: $CP_{1,c} = 0.5Btu/lbmol/F = 0.5 * 1.05506/0.453592 * 1.8kJ/mol/C = 2.0934kJ/mol/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).
Then the liquid phase enthalpy is calculated according to (3.35).

Default: 0.

Example
```
SPECHB
0.0066 0.0052 /
```

This example specifies the second coefficient of the formula of component liquid enthalpy for two components.

6.8.55 SPECHC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the third coefficient $CP_{3,c}$ in the formula below ($kJ/kg/°C/°C/°C$). The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

where the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHA (see 6.8.53), SPECHB (see 6.8.54) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).
In stars data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients $CP_{i,c}$, $i = 1, \ldots, 4$ are specified using the keywords CPL1 / CPL2 / CPL3 / CPL4 (see 8.4.18). tNavigator also uses the keywords $CP_{3,c} =$ SPECHC, $CP_{4,c} =$ SPECHD (see 6.8.56). Default values: $CP_{1,c} = 0.5 Btu/lbmol/F = 0.5 * 1.05506/0.453592 * 1.8kJ/mol/C = 2.0934kJ/mol/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).
Then the liquid phase enthalpy is calculated according to (3.35).

Default: 0.

Example
```
SPECHC
0.00066 0.00052 /
```

This example specifies the third coefficient of the formula of component liquid enthalpy for two components.

6.8.56   SPECHD

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the third coefficient $CP_{4,c}$ in the formula below ($kJ/kg/°C/°C/°C/°C$).
The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

where the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHA, SPECHB (see 6.8.54) (default: 0), T_{ref} is specified using STCOND (see 6.7.7).
   In stars data format **the component liquid enthalpy** is calculated as

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients $CP_{i,c}$, $i = 1, \ldots, 4$ are specified using the keywords CPL1 / CPL2 / CPL3 / CPL4 (see 8.4.18). tNavigator also uses the keywords $CP_{3,c} =$ SPECHC (see 6.8.55), $CP_{4,c} =$ SPECHD. Default values: $CP_{1,c} = 0.5Btu/lbmol/F = 0.5 * 1.05506/0.453592 * 1.8kJ/mol/C = 2.0934kJ/mol/C$, the other coefficients: 0, T_{ref} is specified using TEMR (see 8.4.4).
   Then the liquid phase enthalpy is calculated according to (3.35).

Default: 0.

Example
SPECHD
0.000066 0.000052 /

This example specifies the fourth coefficient of the formula of component liquid enthalpy for two components.

6.8.57 SPECHG

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the first coefficient $CP_{1,c}(kJ/kg/°C)$ in the formula 3.42. The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

Example
```
SPECHG
0.0066 0.0052 /
```

This example specifies the first coefficient of the formula of component gas phase enthalpy for two components.

6.8.58 SPECHH

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the second coefficient $CP_{2,c}$ ($kJ/kg/°C$) in the formula 3.42. The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

Example
```
SPECHH
0.00066 0.00052 /
```

This example specifies the second coefficient of the formula of component gas phase enthalpy for two components.

6.8.59 SPECHI

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the third coefficient $CP_{3,c}$ ($kJ/kg/°C/°C/°C$) in the formula 3.43. The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

Example
```
SPECHI
0.000066 0.000052 /
```

This example specifies the third coefficient of the formula of component gas phase enthalpy for two components.

6.8.60 SPECHJ

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the fourth coefficient $CP_{4,c}$ ($kJ/kg/°C/°C/°C/°C$) in the formula 3.43. The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

Example
```
SPECHJ
0.0000066 0.0000052 /
```

This example specifies the fourth coefficient of the formula of component gas phase enthalpy for two components.

6.8.61   HEATVAPS

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the coefficient $h_{c,G}$ (kJ/kg) (heat of vaporization at temperature STCOND (see 6.7.7)) in the formula 3.42.

The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

Default: 0.

Example
```
HEATVAPS
1200 2100 /
```

This example specifies the coefficient $h_{c,G}$ (heat of vaporization) of the formula of component gas phase enthalpy for two components.

6.8.62 SPECHS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the first coefficient $CP_{1,c}$ in the formula below ($kJ/kg/°C$). The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the component solid phase enthalpy**

$$H_{c,S}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

- in e300 data format the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHS, SPECHT (see 6.8.63) (default: 0), T_{ref} is specified using the keyword STCOND (see 6.7.7).

- in stars data format the coefficients $CP_{1,c}$, $CP_{2,c}$ are specified using the keywords SOLID_CP (see 8.4.14) (default: $CP_{1,c} = 17 kJ/mol/C$, $CP_{2,c} = 0$), T_{ref} is specified using the keyword TEMR (see 8.4.4).

Then the solid phase enthalpy is calculated according to (3.35).

Default: 0.

```
Example
SPECHS
0.52 0.58 /
```

This example specifies the first coefficient of the formula of component solid phase enthalpy for two components.

6.8.63 SPECHT

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword specifies the second coefficient $CP_{2,c}$ in the formula below ($kJ/kg/°C/°C$).
The same number of values as the number of components: N should be entered (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If there are more then one EoS region, the data should be entered for all regions (the number of EoS regions is equal to the 9-th parameter of the keyword TABDIMS (see 6.1.13)). The data for each region should be terminated with a slash /.

In e300 data format **the component solid phase enthalpy**

$$H_{c,S}(T) = CP_{1,c}(T - T_{ref}) + \frac{1}{2}CP_{2,c}(T - T_{ref})^2$$

- in e300 data format the coefficients $CP_{1,c}$ ($kJ/kg/°C$), $CP_{2,c}$ are specified using the keywords SPECHS (see 6.8.62), SPECHT (default: 0), T_{ref} is specified using the keyword STCOND (see 6.7.7).

- in stars data format the coefficients $CP_{1,c}$, $CP_{2,c}$ are specified using the keywords SOLID_CP (see 8.4.14) (default: $CP_{1,c} = 17kJ/mol/C$, $CP_{2,c} = 0$), T_{ref} is specified using the keyword TEMR (see 8.4.4).

Then the solid phase enthalpy is calculated according to (3.35).

Default: 0.

Example
```
SPECHT
0.0052 0.0058 /
```

This example specifies the second coefficient of the formula of component solid phase enthalpy for two components.

6.8.64 TEMPVD

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the table of initial temperature distribution versus depth for each equilibrium region (EQLDIMS (see 6.1.14)). The data should be terminated with a slash /.

Each table row consists of the following parameters:

1. depth,

2. temperature at this depth (C).

Example
```
TEMPVD
2500 180
3600 218
/
```

In this example temperature values are specified at two depth values.

6.8.65   ENPTVT

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
This keyword specifies the temperature variation of the saturation table end points for the saturation table regions. The number of tables can not be greater then the 8-th parameter of the keyword TABDIMS (see 6.1.13). Ecah table shpuld be terminated with a slash /.

The flow of each phase across each grid block is calculated using capillary pressure and relative permeability curves obtained be linearly scaling the tabulated curves between the end points (specified via this keyword ENPTVT (see 6.8.65)).

Each row should contain the following parameters:

1. temperature ($^\circ$C);

2. S_{WL} – minimal value of water saturation S_W;

3. S_{WCR} – maximal (critical) value of water saturation S_W (for which $k_{rW}(S_W) = 0$);

4. S_{WU} – maximal value of water saturation S_W;

5. S_{GL} – minimal value of gas saturation S_G;

6. S_{GCR} – maximal (critical) value of gas saturation S_G (for which $k_{rG}(S_G) = 0$);

7. S_{GU} – maximal value of gas saturation S_G;

8. S_{OWCR} – maximal (critical) value $S_O = 1 - S_W - S_{GL}$, for which $k_{rOW}(S_W) = 0$;

9. S_{OGCR} – maximal (critical) value $S_O = 1 - S_G - S_{WL}$, for which $k_{rOG}(S_G) = 0$.

Parameters 2-9 should be in the range 0.0 to 1.0.

```
Example
ENPTVT
120 0.23 0.21 1.0 0.0 0.05 1.0 0.21 0.24
210 0.25 0.25 1.0 0.0 0.05 1.0 0.21 0.24 /
```

In this example there are saturation table end points for two different temperatures.

6.8.66 ENKRVT

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the tables of relative permeabilities versus temperature. The table should be specified for every saturation table region (the number of saturation table regions is set via the 1-st parameter of TABDIMS (see 6.1.13). Each table should be terminated with a slash /.

Each row should contain the following parameters:

1. temperature ($^{\circ}C$);

2. k_{rWmax} – maximal value of water relative permeability $k_{rW}(S_W)$;

3. k_{rGmax} – maximal value of gas relative permeability $k_{rG}(S_G)$;

4. k_{rOmax} – maximal value of oil relative permeability (functions $k_{rOW}(S_W)$ and $k_{rOG}(S_G)$);

5. k_{rWR} – water relative permeability at crtitical oil saturation ($k_{rW}(1 - S_{OWCR} - S_{GL})$);

6. k_{rGR} – gas relative permeability at critical oil saturation ($k_{rG}(1 - S_{OGCR} - S_{WL})$);

7. k_{rORG} – oil relative permeability at critical gas saturation ($k_{rOG}(S_{GCR})$);

8. k_{rORW} – oil relative permeability at critical water saturation ($k_{rOW}(S_{WCR})$).

Example
```
ENKRVT
120 0.53 1.0 1.0 0.34 1.0 0.7 0.7
220 0.71 1.0 0.89 0.38 1.0 0.7 0.7 /
```

In this example there are relative permeabilities for two different temperatures.

6.8.67   ENPCVT

✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword can only be used if THERMAL (see 6.1.25) is present.
The keyword sets the tables of capillary pressures versus temperature. The table should be specified for every saturation table region (the number of saturation table regions is set via the 1-st parameter of TABDIMS (see 6.1.13). Each table should be terminated with a slash /.

Each row should contain the following parameters:

1. temperature ($°C$);

2. P_{CGmax} – maximum gas capillary pressure $P_{cOG}(S_G)$ ($bars$);

3. P_{CWmax} – maximum water capillary pressure $P_{cOW}(S_W)$ ($bars$).

Example
```
ENPCVT
120 11.4 0.4
220 16.2 4.7 /
```

In this example there are capillary pressures for two different temperatures.

6.8.68 ROCKV

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword can only be used if THERMAL (see 6.1.25) is present.

The keyword sets rock volume in each grid block (m^3). The same number of values should be specified as the number of grid blocks is. The data should be terminated with a slash /.

Formula of grid block volume is in the section 3.25.

If rock volume and pore volume of a grid block are set to zero, this block is inactive.

Example
```
ROCKV
120*1500 200*0 2500*14432 /
```

In this example rock volume of 120 blocks – $1500\,m^3$, of 200 blocks – $0\,m^3$, of 2500 blocks – $14432\,m^3$.

6.9   Initialization section

This section describes initial distribution of saturations and pressures.

There are two ways to specify initial conditions:

- Initial equilibrium, using keyword EQUIL (see 6.9.1) and (for 3-phase models) either PBVD (see 6.9.3) or RSVD (see 6.9.2); user has to specify one of these two keywords unless GOC depth coincides with reference depth in all equilibrium regions ($D_{GOc} = D_{dat}$);

- User defined saturations and pressure distribution, using keywords SWAT (see 6.9.9), SGAS (see 6.9.10) (for 3-phase models), PRESSURE (see 6.9.7) (or PRVD (see 6.9.8)), PBUB (see 6.9.18) or RS (see 6.9.19)(for live oil models), RV (see 6.9.20) or PDEW (see 6.9.21)(for wet gas models).

The keywords specifying initial pressures and saturations (see Initialization section (6.9)):

- RV (see 6.9.20) — initial oil vaporization with wet gas (wet gas);

- PDEW (see 6.9.21) — initial dew point pressure (wet gas);

- RS (see 6.9.19) — initial gas solution in oil (live oil);

- PBUB (see 6.9.18) — initial bubble point pressure (live oil).

There are two keyword groups (1 — wet gas, 2 — live oil). One should use only one keyword from one group.

group 1: RV (see 6.9.20), PDEW (see 6.9.21)

group 2: RS (see 6.9.19), PBUB (see 6.9.18)

One can see different keyword combinations in Property section (6.5) (PVT properties, permeabilities, initial pressures and saturations).

6.9.1 EQUIL

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies for each equilibrium region properties used in initial conditions computations. The following parameters are to be specified:

1. reference depth, D_{dat};

2. pressure on reference depth, P_{dat};

3. depth of water-oil contact (WOC), D_{WOc};

4. capillary pressure at water-oil contact, P_{WOc};

5. depth of gas-oil contact (GOC), D_{GOc};

6. capillary pressure at gas-oil contact, P_{GOc};

7. parameter regulating table RSVD (see 6.9.2) (or PBVD (see 6.9.3)) usage for initialization calculation. If set to positive integer, the noted table will be used to calculate oil bubble point pressure (under condition that solution gas oil ratio can't exceed saturated solution gas oil ratio value for current pressure). If parameter is set to negative or zero value, tables RSVD (see 6.9.2) and PBVD (see 6.9.3) will be ignored, in this case for live oil it's necessary that GOC corresponds with datum depth: $D_{GOc} = D_{dat}$. For dead oil case (without solution gas) this parameter will be ignored;

8. parameter regulating table RVVD (see 6.9.4) (or PDVD (see 6.9.5)) usage for initialization calculation. If set to positive integer, the noted table will be used to calculate gas dew point pressure (under condition that vaporized oil concentration can't exceed saturated vaporized oil concentration for current pressure). If parameter is set to negative or zero value, tables RVVD (see 6.9.4) and PDVD (see 6.9.5) will be ignored, in this case for wet gas it's necessary that GOC corresponds with datum depth: $D_{GOc} = D_{dat}$. For dry gas case (without vaporized oil) this parameter will be ignored;

9. integer defining accuracy of equilibrium calculation; if set to zero, equilibrium will be calculated for block centers. This solution will be steady-state. Otherwise (value other than zero) grid blocks will be divided into thin layers (with thickness equal to 1thinnest reservoir active block, but not less than 0.0011 m) and equilibrium will be calculated for each one of those layers. Resulting saturations will be calculated as average values.

The number of lines specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each line must end with slash (/).

Default:

- For water-oil capillary pressure default value is zero: $P_{WOc} = 0$;

- For gas-oil capillary pressure default value is zero: $P_{GOc} = 0$.

Notes:

- For depths specified the following relation must be fulfilled: $D_{GOc} \leq D_{WOc}$.

- If D_{dat} is above GOC, pressure P_{dat} will correspond to gas phase pressure on this depth; if D_{dat} is below WOC, it corresponds to water phase, otherwise – oil phase.

- For 3-phase models, when $D_{GOc} = D_{dat}$, oil bubble point pressure on GOC and below will be taken equal to P_{dat}, and above – to grid block pressure. If $D_{GOc} <> D_{dat}$, to provide data for oil bubble point pressure calculation user should specify either RSVD (see 6.9.2) or PBVD (see 6.9.3).

- For each equilibrium region grid blocks from this region should not have different numbers of PVT regions (PVTNUM (see 6.4.1)).

- Gas-oil contact D_{GOc} may be less then reservoir depth (TOPS (see 6.2.5)), that means mobile gas phase absence. Gas-oil contact D_{GOc} may be greater then reservoir bottom depth, that means that only gas is mobile in the model.

- Water-oil contact D_{WOc} may be less then reservoir depth (TOPS (see 6.2.5)), that means that only water is mobile in the model. Water-oil contact D_{WOc} may be greater then reservoir bottom depth, that means mobile water phase absence.

```
Example
EQLDIMS
3 /
...
EQUIL
2300 200 2500 0.1 2300 0.001 /
2310 205 2520 0.05 2310 0.0 /
2305 210 2510 1* 2305 1* /
```

In this example the data for three regions are specified. Default values for capillary pressures in the third region are zero values. Since GOC coincides with D_{dat} for all three regions, there's no need to define RSVD (see 6.9.2) or PBVD (see 6.9.3).

6.9.2 RSVD

✓	tNavigator	✓	eclipse 300	☐	MORE Roxar
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword specifies for each equilibrium region initial solution gas ratio distribution, used in initial conditions computations in case of equilibrium initialization using EQUIL (see 6.9.1). If in a 3-phase live oil model all equilibrium regions have GOC equal to D_{dat}, there is no need to define RSVD (see 6.9.2) or PBVD (see 6.9.3). Otherwise either RSVD or PBVD (see 6.9.3) should be specified.

The following parameters are to be specified:

- depth;

- solution gas ratio value at this depth.

The number of tables specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each table should end with slash (/). Between depth points dissolved gas-oil ratio will be interpolated linearly.

```
Example
EQLDIMS
2 /
...
EQUIL
2300 200 2500 0.1 2300 0.001 1 /
2310 205 2520 0.05 2360 0.0 1 /
/
RSVD
2000 60
2500 68
3000 73.5
/
2000 100
3000 200
/
```

In this example the data for two regions are specified. Since in the first region GOC coincides with reference depth D_{dat}, first RSVD table will not be used, because bubble point pressure on GOC and below will be set equal to datum pressure, 200.

In the second equilibration region GOC depth differs from D_{dat}, so second (2-line) RSVD table will be used to calculate oil bubble point pressure.

6.9.3 PBVD

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword specifies for each equilibrium region initial bubble point pressure distribution, used in initial conditions computations in case of equilibrium initialization using EQUIL (see 6.9.1). If in a 3-phase live oil model all equilibrium regions have GOC equal to D_{dat}, there is no need to define RSVD (see 6.9.2) or PBVD (see 6.9.3). Otherwise either PBVD or RSVD (see 6.9.2) should be specified.

The following parameters are to be specified:

- depth;

- bubble point pressure value at this depth.

The number of tables specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each table should end with slash (/). Between depth points bubble point pressure will be interpolated linearly.

```
Example
EQLDIMS
2 /
...
EQUIL
2300 200 2500 0.1 2400 0.001 1 /
2310 205 2520 0.05 2310 0.0 1 /

PBVD
2000 250
2500 310
3000 350
/
2000 200
3000 300
/
```

In this example the data for two regions are specified. Since in the second region GOC coincides with reference depth D_{dat}, second PBVD table will not be used, because bubble point pressure on GOC and below will be set equal to datum pressure, 205.

In the first equilibration region GOC depth differs from D_{dat}, so first (3-line) PBVD table will be used to calculate oil bubble point pressure.

6.9.4 RVVD

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies for each equilibrium region initial vaporized oil concentration distribution, used in initial conditions computations in case of equilibrium initialization using EQUIL (see 6.9.1). If in a 3-phase wet gas model all equilibrium regions have GOC equal to D_{dat}, there is no need to define RVVD (see 6.9.4) or PDVD (see 6.9.5). Otherwise either RVVD or PDVD (see 6.9.5) should be specified.

The following parameters are to be specified:

- depth;

- vaporized oil concentration value at this depth.

The number of tables specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each table should end with slash (/). Between depth points dissolved gas-oil ratio will be interpolated linearly.

```
Example
EQLDIMS
2 /
...
EQUIL
2300 200 2500 0.1 2300 0.001 0 1 /
2310 205 2520 0.05 2360 0.0 0 1 /
/
RVVD
2000 0.00060
2500 0.00068
3000 0.000735
/
2000 0.000100
3000 0.000200
/
```

In this example the data for two regions are specified. Since in the first region GOC coincides with reference depth D_{dat}, first RVVD table will not be used, because dew point pressure on GOC and below will be set equal to datum pressure, 200.

In the second equilibration region GOC depth differs from D_{dat}, so second (2-line) RVVD table will be used to calculate gas dew point pressure.

6.9.5 PDVD

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies for each equilibrium region initial dew point pressure distribution, used in initial conditions computations in case of equilibrium initialization using EQUIL (see 6.9.1). If in a 3-phase wet gas model all equilibrium regions have GOC equal to D_{dat}, there is no need to define RVVD (see 6.9.4) or PDVD (see 6.9.5). Otherwise either PDVD or RVVD (see 6.9.4) should be specified.

The following parameters are to be specified:

- depth;

- dew point pressure value at this depth.

The number of tables specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each table should end with slash (/). Between depth points bubble point pressure will be interpolated linearly.

```
Example
EQLDIMS
2 /
...
EQUIL
2300 200 2500 0.1 2400 0.001 0 1 /
2310 205 2520 0.05 2310 0.0 0 1 /

PDVD
2000 250
2500 310
3000 350
/
2000 200
3000 300
/
```

In this example the data for two regions are specified. Since in the second region GOC coincides with reference depth D_{dat}, second PDVD table will not be used, because dew point pressure on GOC and below will be set equal to datum pressure, 205.

In the first equilibration region GOC depth differs from D_{dat}, so first (3-line) PDVD table will be used to calculate gas dew point pressure.

6.9.6   THPRES

	tNavigator		eclipse 300		MORE Roxar
√		√		☐	
√	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword is used to set threshold pressure value for flow between equilibration regions. If pressure difference between cells of specified regions is less than threshold value, there will be no flow between those cells. Otherwise flow between cells will be calculated basing on pressure difference reduced by threshold value. By default, threshold value is equal to 0.

See also keyword THPRESFT (see 6.2.29).

Keyword is followed by a 3-column table, each line should end with slash /. Last table line should contain only slash symbol (/).

The following data should be specified:

- number of first equilibration region;

- number of second equilibration region;

- threshold pressure value for flow between those regions.

Default: threshold pressure equal to 0.

Note: if two regions are indicated several times (in any order), last value of threshold pressure will be used in calculations.

```
Example
THPRES
1 2  10.0 /
2 1  5.0 /
1 3  15.1 /
3 1  15.1 /
2 3  7.0 /
1 4  /
4 1  /
/
```

In this example threshold pressure in any direction will be equal to 5 for regions 1 and 2 (last indicated value), 15.1 for regions 1 and 3, 7 for regions 2 and 3, default (zero) for regions 1 and 4.

6.9.7 PRESSURE

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword specifies initial pressures. The same number of values as number of grid blocks must be specified. This is initial value of p_O in 2.3.

Default: none

```
Example
DIMENS
5 5 1
/
...
PRESSURE
25*235
/
```

This example sets equal pressures = 235 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
5 3 2
/
...
PRESSURE
235 235 235 235 235 235 235 235
235 235 235 235 235 235 235
250 250 250 250 250 250 250 250
250 250 250 250 250 250 250
/
```

This example specifies pressure equal to 235 in the first layer of a 5x3x2 two-layer grid, and pressure equal to 250 in the second layer.

```
Example
DIMENS
5 3 2
/
...
PRESSURE
15*235 15*250
/
```

This example is equivalent to the previous one, but uses the short form with asterisks.

6.9.8 PRVD

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies initial pressure for each depth level. It can be used to specify initial pressure distribution instead of PRESSURE (see 6.9.7) keyword.

Table should consist of at least two lines. The following parameters are to be specified on each line:

- depth;

- pressure value at this depth.

The number of tables specified is equal to number of equilibrium regions defined in EQLDIMS (see 6.1.14). Each table should end with slash (/). Between depth points pressure will be interpolated linearly. Extrapolation outside specified pressure interval will also be linear.

```
Example
EQLDIMS
2 /
PRVD
2000 250
2500 310
3000 350
/
2000 200
3000 300
/
```

In this example the data for two regions are specified.

6.9.9 SWAT

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies initial water saturations. The same number of values as the number of grid blocks must be specified. This is initial value of S_W in 2.3.

If only water saturation is specified, gas saturation is defaulted to zero and initial oil saturation is calculated automatically according to formula $S_O = 1 - S_W$.

Default: none

```
Example
DIMENS
5 5 1
/
...
SWAT
25*0.25
/
```

This example sets initial water saturation equal to 0.25 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
5 3 2
/
...
SWAT
0.25 0.25 0.25 0.25 0.25 0.25 0.25 0.25
0.25 0.25 0.25 0.25 0.25 0.25 0.25
0.50 0.50 0.50 0.50 0.50 0.50 0.50 0.50
0.50 0.50 0.50 0.50 0.50 0.50 0.50
/
```

This example sets different water saturations in two layers of a 5x3x2 grid: 0.25 in the upper layer and 0.50 in the lower layer.

```
Example
DIMENS
5 3 2
/
...
SWAT
15*0.25 15*0.50
/
```

This example is equivalent to the previous one, but it uses short form with asterisks.

6.9.10 SGAS

	tNavigator		eclipse 300		MORE Roxar
✓		✓		□	
✓	eclipse 100	□	CMG imex	□	CMG stars

This keyword specifies initial gas saturations. The same number of values as the number of grid blocks must be specified. This is initial value of S_G in 2.3.

If SGAS (see 6.9.10) is specified, then gas is considered to present in the system and initial oil saturation is calculated according to formula $S_O = 1 - S_G - S_W$

Default: none

```
Example
DIMENS
5 5 1
/
...
SGAS
25*0.75
/
```

This example sets initial gas saturation equal to 0.75 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
5 3 2
/
...
SGAS
0.75 0.75 0.75 0.75 0.75 0.75 0.75 0.75
0.75 0.75 0.75 0.75 0.75 0.75 0.75
0.50 0.50 0.50 0.50 0.50 0.50 0.50 0.50
0.50 0.50 0.50 0.50 0.50 0.50 0.50
/
```

This example sets different gas saturations in two layers of a 5x3x2 grid: 0.75 in the upper layer and 0.50 in the lower layer.

```
Example
DIMENS
5 3 2
/
...
SGAS
15*0.75 15*0.50
/
```

This example is equivalent to the previous one, but it uses short form with asterisks.

6.9.11 SOIL

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies initial oil saturations. The same number of values as the number of grid blocks must be specified. This is initial value of S_O in 2.3.

If only SOIL (see 6.9.11) is specified, then gas is considered to be apsent in the system and initial water saturation is calculated according to formula $S_W = 1 - S_O$

Default: none

Example
```
DIMENS
5 5 1
/
...
SOIL 25*0.21
/
```

This example sets initial oil saturation equal to 0.21 in all 25 grid blocks (grid dimensions are 5x5x1).

Example
```
DIMENS
5 3 2
/
...
SOIL
0.21 0.21 0.21 0.21 0.21 0.21 0.21 0.21
0.21 0.21 0.21 0.21 0.21 0.21 0.21
0.50 0.50 0.50 0.50 0.50 0.50 0.50 0.50
0.50 0.50 0.50 0.50 0.50 0.50 0.50
/
```

This example sets different oil saturations in two layers of a 5x3x2 grid: 0.21 in the upper layer and 0.50 in the lower layer.

Example
```
DIMENS
5 3 2
/
...
SOIL
15*0.21 15*0.50
/
```

This example is equivalent to the previous one, but it uses short form with asterisks.

6.9.12 SSOLID

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the initial solid saturation. The same number of values as the number of blocks should be specified. The data should be terminated with a slash /.

Default: not specified.

Example
```
SSOLID
1500*0.087
```

In this example 1500 dgid blocks have the same initial solid saturation 0.087.

6.9.13 SMF

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword specifies cell initial solid composition. The same number of values as the product of grid blocks NX * NY * NZ (DIMENS (see 6.1.12)) and the number of components (COMPS (see 6.7.3)) should be specified. The data should be terminated with a slash /.

Default: not specified.

Example
```
DIMENS
5 10 2 /
COMPS
5 /
...
SMF
100*0.
100*0.
100*0.
100*0.
100*1.
/
```

This example sets initial solid composition for 100 grid blocks and 5 components.

6.9.14   XMF

√ **tNavigator**	√ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the initial oil composition for each grid block. One should specify the same number of values as the product of numbers of grid blocks NX * NY * NZ (DIMENS (see 6.1.12)) and the number of components (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If the sum of values for any grid block isn't 1 then the values are normalized so that the error is less than 10 percents. If the error if greater than 10 percents the message of data initializatin error will occur.

Default: none

```
Example
DIMENS
5 10 2 /
COMPS
5 /
...
XMF
100*0.5
100*0.3
100*0.2
100*0.
100*0.
/
```

This example sets initial oil composition for 100 grid blocks and 5 components.

6.9.15   YMF

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets the initial gas composition for each grid block. One should specify the same number of values as the product of numbers of grid blocks NX * NY * NZ (DIMENS (see 6.1.12)) and the number of components (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If the sum of values for any grid block isn't 1 then the values are normalized so that the error is less than 10 percents. If the error if greater than 10 percents the message of data initializatin error will occur.

Default: none

```
Example
DIMENS
5 10 2 /
COMPS
5 /
...
YMF
100*1
100*0
100*0
100*0
100*0
/
```

This example sets initial gas composition for 100 grid blocks and 5 components.

6.9.16 ZMF

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the initial total composition for each grid block (values $z'_c = N_c/N'_{tot}$, $c = 2, \ldots, n'_c$, $N'_{tot} = \sum\limits_{c=2}^{n'_c} N_c$). One should specify the same number of values as the product of numbers of grid blocks NX * NY * NZ (DIMENS (see 6.1.12)) and the number of components (COMPS (see 6.7.3)). The data should be terminated with a slash /.

If the sum of values for any grid block isn't 1 then the values are normalized so that the error is less than 10 percents. If the error if greater than 10 percents the message of data initializatin error will occur.

Default: none

```
Example
DIMENS
5 10 2 /
COMPS
5 /
...
ZMF
100*0.5
100*0.2
100*0.2
100*0.06
100*0.04
/
```

This example sets initial total composition for 100 grid blocks and 5 components.

6.9.17 TEMPI

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the initial temperature ($°$C) in each grid block. The same number of values as the number of grid blocks must be specified. The data should be terminated with a slash `/`.

Default: none

```
Example
DIMENS
10 10 5 TEMPI
250*100 250*120
/
```

This example sets the initial temperature in the reservoir: 100$°$C in first 250 grid blocks and 120$°$C in next 250 grid blocks.

6.9.18   PBUB

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword specifies initial bubble point pressure for undersaturated reservoir. The same number of values as number of grid blocks must be specified. The data should be terminated by a slash /.

This is initial value of $R_{G,O}$ in 2.3.

This keyword is an alternative to the keyword RS (see 6.9.19) (only one of them should be specified).

Default: none

Note: The following rules apply for initializing the reservoir with user specified initial bubble point pressure:

- If the gas saturation in some grid block is non zero, then the specified bubble point pressure for this grid block is ignored and it is reset to block pressure in this grid block;

- If specified bubble point pressure in some block exceeds the initial pressure in this grid block the bubble point pressure for this grid block is reset to initial block pressure;

```
Example
DIMENS
5 5 1
/
...
PBUB
25*235
/
```

This example sets equal initial bubble point pressures = 235 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
5 3 2
/
...
PBUB
235 235 235 235 235 235 235 235
235 235 235 235 235 235 235
250 250 250 250 250 250 250 250
250 250 250 250 250 250 250
/
```

This example specifies initial bubble point pressure equal to 235 in the first layer of a 5x3x2 two-layer grid, and equal to 250 in the second layer.

```
Example
DIMENS
5 3 2
/
...
PBUB
15*235 15*250
/
```

This example is equivalent to the previous one, but uses the short form with asterisks.

6.9.19   RS

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies initial gas solution in oil for live oil model. The same number of values as number of grid blocks must be specified.

The data should be terminated by a slash /.

This keyword is an alternative to the keyword PBUB (see 6.9.18) (only one of them should be specified).

```
Example
DIMENS
4 4 1
/
...
RS
16*0.003
/
```

This example sets equal initial gas solution in oil = 0.003 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
3 3 2
/
...
RS
.235 .235 .235 .235 .235 .235 .235 .235 .235
.250 .250 .250 .250 .250 .250 .250 .250 .250
/
```

This example specifies initial gas solution in oil equal to .235 in the first layer of a 5x3x2 two-layer grid, and equal to .0250 in the second layer.

```
Example
DIMENS
3 3 2
/
...
RS
9*.235 9*.250
/
```

This example is equivalent to the previous one, but uses the short form with asterisks.

6.9.20 RV

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword specifies initial oil vaporisation in gas for wet gas model. The same number of values as number of grid blocks must be specified. This is initial value of $R_{O,G}$ in 2.3.

The data should be terminated by a slash /.

This keyword is an alternative to the keyword PDEW (see 6.9.21) (only one of them should be specified).

Default: none

```
Example
DIMENS
5 5 1
/
...
RV
25*0.0001
/
```

This example sets equal initial oil vaporisation in gas values = 0.0001 in all 25 grid blocks (grid dimensions are 5x5x1).

```
Example
DIMENS
5 3 2
/
...
RV
.0235 .0235 .0235 .0235 .0235 .0235 .0235 .0235
.0235 .0235 .0235 .0235 .0235 .0235 .0235
.0250 .0250 .0250 .0250 .0250 .0250 .0250 .0250
.0250 .0250 .0250 .0250 .0250 .0250 .0250
/
```

This example specifies initial oil vaporisation equal to .0235 in the first layer of a 5x3x2 two-layer grid, and equal to .0250 in the second layer.

Example
```
DIMENS
5 3 2
/
...
RV
15*.0235 15*.0250
/
```

This example is equivalent to the previous one, but uses the short form with asterisks.

6.9.21 PDEW

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword specifies initial bubble point pressure for black oil model. The same number of values as number of grid blocks must be specified. The data should be terminated by a slash /.

This keyword is an alternative to the keyword RV (see 6.9.20) (only one of them should be specified).

Default: none

```
Example
DIMENS
6 5 2
/
...
PDEW
60*2800
/
```

This example sets equal initial bubble point pressure = 2800 in all 60 grid blocks (grid dimensions are 6x5x2).

```
Example
DIMENS
7 5 2
/
...
PDEW
35*2350 35*1250
/
```

This example specifies initial bubble point pressure to 2350 in the first layer of a 5x3x2 two-layer grid, and equal to 1250 in the second layer.

6.9.22　DATUM

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the datum depth for calculations of depth corrected pressures. If this keyword isn't specified the depth entered by 1-st parameter of EQUIL (see 6.9.1) for 1-st equilibrium region will be used.

Example
```
DATUM 3000
/
```

6.9.23   DATUMR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets the datum depth for calculations of depth corrected pressures for each FIP region.

One should spesify the same number of values as the number of FIP regions (5-th parameter of TABDIMS (see 6.1.13)).

```
Example
TABDIMS 3 2 2* 4
DATUMR 3000 4200 5100 2800
/
```

This example sets datum depths for 4 FIP regions.

6.9.24 TBLK

☑	tNavigator	☑	eclipse 300	☐	MORE Roxar
☑	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies the initial concentration of a tracer in each grid block The keyword should be followed by real numbers, specifying the initial tracer concentration. The same number of values as the number of grid blocks should be specified. Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y-axis index, followed by the Z-axis index. The data should be terminated with a slash /.

The keyword may be entered in two formats: e100 and e300.
The keyword in e100 format:

1. characters 1-4 should be – TBLK;

2. character 5 – letter F (for the tracer whose associated phase can only exist in the free state) or letter S (for the tracer whose associated phase can only exist in the solution state). If a letter S is specified, this keyword is ignored. If the letter F is specified, this letter is ignored, the keyword will be read in e300 format;

3. other characters – name of the tracer whose concentration is initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

The keyword in e300 format:

1. characters 1-4 should be – TBLK;

2. other characters – name of the tracer whose concentration is initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

```
Example
TRACER
A WAT /
B WAT /
/

...

TBLKFA
2600*1 /

TBLKFB
2600*0. /
```

In this example there are two tracers A and B, associated with water phase. Tracer A has initial concentration 1 in all grid blocks, tracer B – 0 in all grid blocks.

6.9.25 TNUM

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies tracer concentration region numbers for each grid block. For each tracer concentration region the keyword TVDP (see 6.9.26) specifies the table of initial tracer concentration with respect to depth.

The keyword may be entered in two formats: e100 and e300.
The keyword in e100 format:

1. characters 1-4 should be – TNUM;

2. character 5 – letter F (for the tracer whose associated phase can only exist in the free state) or letter S (for the tracer whose associated phase can only exist in the solution state). If a letter S is specified, this keyword is ignored. If the letter F is specified, this letter is ignored, the keyword will be read in e300 format;

3. other characters – name of the tracer whose tracer concentration regions are initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

The keyword in e300 format:

1. characters 1-4 should be – TNUM;

2. other characters – name of the tracer whose tracer concentration regions are initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

The keyword should be followed by one integer for every grid block, specifying the initial tracer concentration region number to which it belongs. The region number should not be grater than the 4-th parameter of the keyword EQLDIMS (see 6.1.14). Grid blocks are ordered with the X-axis index cycling fastest, followed by the Y-axis index, followed by the Z-axis index. The data should be terminated with a slash /.

Example
```
TNUMFB
32*1 32*2
/

...

TVDPFB
0 0.1
100 0.3
200 0.5 /
0 0.2
100 0.4
200 0.6 /
/
```

In this example tracer B has 2 initial concentration regions (specified via TNUM (see 6.9.25)): first 32 blocks belong to 1-nd region, next 32 blocks – 2-nd region. These two regions have two corresponding tables of initial tracer concentration with respect to depth (specified via TVDP (see 6.9.26)).

6.9.26   TVDP

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies tables of initial tracer concentration with respect to depth for each tracer. Tracer concentration region numbers for each grid block are specified via the keyword TNUM (see 6.9.25).

The keyword may be entered in two formats: e100 and e300.
The keyword in e100 format:

1. characters 1-4 should be – TVDP;

2. character 5 – letter F (for the tracer whose associated phase can only exist in the free state) or letter S (for the tracer whose associated phase can only exist in the solution state). If a letter S is specified, this keyword is ignored. If the letter F is specified, this letter is ignored, the keyword will be read in e300 format;

3. other characters – name of the tracer whose table of initial tracer concentration with respect to depth is initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

The keyword in e300 format:

1. characters 1-4 should be – TVDP;

2. other characters – name of the tracer whose table of initial tracer concentration with respect to depth is initialized (name is specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters).

The keyword should be followed by table of initial tracer concentration with respect to depth. For each concentration region (entered via TNUM (see 6.9.25)) should be specified one table. Each table should be terminated with a slash /. All tables should be terminated with a final slash /.

Each table row should consist of the following parameters (the number of lines should not be greater than the 5-th parameter of the keyword EQLDIMS (see 6.1.14)):

1. depth (m);

2. the corresponding initial tracer concentration at this depth (the concentration should be not negative).

```
Example
TNUMFB
32*1 32*2
/

...

TVDPFB
0 0.1
100 0.3
200 0.5 /
0 0.2
100 0.4
200 0.6 /
/
```

In this example tracer B has 2 initial concentration regions (specified via TNUM (see 6.9.25)): first 32 blocks belong to 1-nd region, next 32 blocks – 2-nd region. These two regions have two corresponding tables of initial tracer concentration with respect to depth (specified via TVDP (see 6.9.26)).

6.9.27 ROCKSALT

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword is used to specify initial mass of reservoir salt that can be dissolved. The same number of values as the number of grid blocks should be spesified. The data should be terminated with a slash /.

This keyword can be used only if the Brine option is active (the keyword BRINE (see 6.1.33)). The dissolution rate constant ϑ_{salt} (the keyword SALTPROP (see 6.5.26)) should also be specified to simulate dissolution process.

```
Example
DIMENS
5 5 1
/

...

ROCKSALT
25*20000
/
```

In this example initial mass of reservoir salt is the same for all grid blocks - $20000(kg)$.

6.9.28 SALT

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword should be followed by one real number for every grid block specifying the initial salt concentration (kg/m^3). The number of values should be equal to the number of grid blocks. Grid blocks should be ordered with the X axis index cycling fastest, followed by the Y axis index and finally the Z axis index. The data should be terminated with a slash /.

The keyword should be used when the initial state has been set by enumeration (keywords PRESSURE (see 6.9.7), RS (see 6.9.19), RV (see 6.9.20), SGAS (see 6.9.10) and SWAT (see 6.9.9)). For a run initialized by equilibration EQUIL (see 6.9.1), the keyword SALTVD (see 6.9.29) should be used instead of SALT (see 6.9.28) (salt concentration versus depth for equilibration).

```
Example
50*0
50*0.75
50*2.6
50*9.2
50*21.0 /
```

6.9.29 SALTVD

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets salt concentration versus depth for equilibration for each equilibration region.

The keyword should be used when the initial state has been set by equilibration EQUIL (see 6.9.1). For a run initialized by enumeration (keywords PRESSURE (see 6.9.7), RS (see 6.9.19), RV (see 6.9.20), SGAS (see 6.9.10) and SWAT (see 6.9.9)). the keyword SALT (see 6.9.28) should be used instead of SALTVD (see 6.9.29).

The data consists of the tables. Number of tables should be equal to the number of equilibration region (first parameter of the keyword EQLDIMS (see 6.1.14)). Each table should be terminated with a slash /.

One table row consists of:

1. depth m;

2. the corresponding value of salt concentration kg/m^3.

```
Example
SALTVD
1000.0 144.0
2000.0 144.0
/
1000.0 144.0
2000.0 144.0
/
```

In this example there are 2 tables of таблицы salt concentration versus depth. In both tables the value of salt concentration is the same at all specified depths and is equal to 144 kg/m^3.

6.10 Schedule section

Keywords in this section are interpreted according to sequence in which they are specified in user file, so user should first specify well, then its perforations and then its schedule.

Well is specified by keyword (WELSPECS (see 6.10.2)) and its perforated intervals by keyword (COMPDAT (see 6.10.6)).

To specify wells in local grids (section 2.21) the following keywords should be used:

- WELSPECL (see 6.10.3) – introduces a new well, defining information on its name and coordinates in local grids (LGR). WELSPECL must be used in place of WELSPECS (see 6.10.2) to set the general specification data for wells in local refined grids.

- COMPDATL (see 6.10.7) – defines well completions in local grids (LGR). COMPDATL (see 6.10.7) must be used in place of COMPDAT (see 6.10.6) to specify the connection data for wells in local refined grids.

- COMPLMPL (see 6.10.9) (SCHEDULE section) – lumps connections together into completions to provide realization of simultaneous actions for wells in local grids (LGR). COMPLMPL (see 6.10.9) must be used in place of COMPLUMP (see 6.10.8) to lump connections together into completions to provide realization of simultaneous actions for wells in local refined grids.

- WPIMULTL (see 6.10.11) (SCHEDULE section) – multiplies well connection transmissibility factors by specified value for wells in local grids (LGR). WPIMULTL (see 6.10.11) must be used in place of WPIMULT (see 6.10.10) to multiply well connection transmissibility factors by specified value for wells in local refined grids.

- WFRACL (see 6.10.42) (SCHEDULE section) – specifies the hydraulic fracture for wells in local refined grids (LGR). WFRACL (see 6.10.42) must be used in place of WFRAC (see 6.10.41) to specify the hydraulic fracture for wells in local refined grids.

- WFRACPL (see 6.10.44) (SCHEDULE section) – specifies the hydraulic fracture for wells in local refined grids (LGR) in graphical interface. WFRACPL (see 6.10.44) must be used in place of WFRACP (see 6.10.43) to specify the hydraulic fracture for wells in local refined grids in graphical interface.

- COMPFRACL (see 6.10.46) (SCHEDULE section) – specifies the hydraulic fracture for connection in the grid layer for wells in local refined grids (LGR). COMPFRACL (see 6.10.46) must be used in place of COMPFRAC (see 6.10.45) to specify the hydraulic fracture for connection in the grid layer for wells in local refined grids.

In order to set well schedule it is necessary to specify well name (or number) and name of group it belongs to.

Either fully qualified well name or masked named may be used. In the latter case the operation desired will be applied to subset of wells with names fitted to mask. In the well

mask along with usual alphabetic and numeric symbols the following special symbols may be used:

- * — corresponds to 0 or more any symbols;

- ? — corresponds to any single symbol;

- [n-m] — corresponds to any single symbol with code in the range from n to m.

Examples of well name specifications:

- P123 — specifies single well with name P123;

- * — specifies all wells;

- P* — specifies all wells with names starting from letter P;

- P1?3 — specifies all wells with 4 letter names and 2 first letters are P1 and the last is 3;

- P1[1-3]3 — specifies wells with names P113, P123, P133 (if they are exist);

Mask as well name cannot be used for new well introduction, but the only for changing of work mode for existing wells.

To set parameter value equal to default, put asterisk *. If several successive parameters are equal to default values, you can write N*, here N is number of parameters for which program will take default values.

For any keyword in this section (except TSTEP and DATES) the optional date may be specified. This date may be specified for the whole keyword block, for example,

---

Example
```
WCONPROD 1 JAN 2003
...
```

---

as well as for any entry in the keyword block, for example,

Example
```
COMPDAT
1043 14 10 1 2 OPEN 8*                21 MAY 2003 /
W13  16 13 1 3 OPEN 3* 20 1 1* Z 1*   21 JUN 2003 /
...
```

If both dates are specified (for whole block and for record in the block) the date for record is used.

Traditional monotonic schedule section is also supported. If optional dates are specified then monotonic section will be reconstructed and new dates will be inserted.

6.10.1 WELSOMIN

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets minimum oil saturation in a block for automatic opening of well connections (well working mode AUTO in WCONPROD (see 6.10.12)). If the limit WELSOMIN isn't reached connections can be opened manually (using the keyword COMPDAT (see 6.10.6)). If the keyword isn't specified well connections (well working mode AUTO), will be opened automatically for any block oil saturation.

The data should be terminated with a slash /.

Example
```
WELSOMIN
0.2 /
```

6.10.2 WELSPECS

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword introduces a new well, defining information on its name and coordinates. A set of data for every well should be terminated with a slash /.

The following data should be specified:

1. well name (or number),

2. name of the group to which this well belongs,

3. bottom hole or well head coordinates in X direction (IW);

4. bottom hole or well head coordinates in Y direction (JW); those are i and j in 5.3.1; these coordinates will be later used as default perforation coordinates in COMPDAT (see 6.10.6),

5. reference depth for bottom hole pressure.
   Recommended location - first perforated interval depth (m),

6. preferred phase for well: OIL, WATER, GAS, LIQ. This data is used to determine the worst offending well or connection for closure when a group production rate limit is exceeded GCONPROD (see 6.10.23) (7-th parameter).
   This data is also used to define the productivity/injectivity index of well or potential flow rate (parameter 7 of this keyword)

7. drainage radius productivity/injectivity index calculations (m) (IGNORED) this is an Eclipse compatibility field.
   Productivity/injectivity index is calculated from a steady-state relationship and is printed in the well reports.
   If 0 is specified, then is used the pressure equivalent radius of the grid blocks containing the well connections. The productivity index is equal ot the sum over the connections of the product of the connection factor and the local mobility of the preferred phase.
   If the value is negative, the well's potential flow rate is printed in the well reports instead of the productivity index. (The well's potential rate is the flow rate it would achieve in the absence of any rate constraint, limited only by its BHP and THP constrait),

8. flag of a special inflow equation to model the flow of gas between the completed grid blocks and the well completions: STD (or NO) – the standart inflow equations will be used, GPP – generalized pseudo-pressure equation (this option may be used by gas condensate producers – the keyword PICOND (see 6.10.64)). Generalized pseudo-pressure equation alters both the gas and oil mobilities and takes accounts also of the effects of condensate dropout,

9. well can be automatically shut or stopped if a well or group economic limit or group flow limit is violated: CodeSTOP - stop well above formation, SHUT - isolate well from the formation,

10. crossflow ability flag: YES - crossflow allowed in the well, NO - crossflow not allowed in the well, connection is a one-way valve, which prevents reverse drawdown flow.

   Default:

   • well group name: FIELD,

   • reference depth for bottom hole pressure - first perforated interval depth,

   • drainage radius – 0,

   • flag of a special inflow equation – STD,

   • automatically shut or stopped if a well or group economic limit or group flow limit is violated – SHUT,

   • crossflow ability flag – YES.

Example
```
DIMENS
25 20 3
...
WELSPECS
1043 G1 14 10 5* NO/
W13  1* 16 13 /
/
```

Here two wells are defined: well 1043, from group G1, it's bottom hole situated at $X = 14$ and $Y = 10$, crossflow not allowed in the well; and well W13, from group FIELD, bottom hole situated at $X = 16$ and $Y = 13$.

6.10.3 WELSPECL

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword introduces a new well, defining information on its name and coordinates in local grids (LGR) – section 2.21.

WELSPECL must be used in place of WELSPECS (see 6.10.2) to set the general specification data for wells in local refined grids. The keyword data is similar to that for WELSPECS (see 6.10.2), except there is an additional item at parameter 3 which gives the name of the local grid refinement in which the well is located. The connection data for these wells must be specified using the keyword COMPDATL (see 6.10.7) instead of COMPDAT (see 6.10.6).

A set of data for every well should be terminated with a slash /.
The following data should be specified:

1. well name (or number),

2. name of the group to which this well belongs,

3. name of the local refined grid in which the well is located (CARFIN (see 6.2.61)),

4. bottom hole or well head coordinates in X direction (IW);

5. bottom hole or well head coordinates in Y direction (JW); those are i and j in 5.3.1; these coordinates will be later used as default perforation coordinates in COMPDAT (see 6.10.6),

6. reference depth for bottom hole pressure.
   Recommended location - first perforated interval depth (m),

7. preferred phase for well: OIL, WATER, GAS, LIQ. This data is used to determine the worst offending well or connection for closure when a group production rate limit is exceeded GCONPROD (see 6.10.23) (7-th parameter).
   This data is also used to define the productivity/injectivity index of well or potential flow rate (parameter 7 of this keyword)

8. drainage radius productivity/injectivity index calculations (m) (IGNORED) this is an Eclipse compatibility field.
   Productivity/injectivity index is calculated from a steady-state relationship and is printed in the well reports.
   If 0 is specified, then is used the pressure equivalent radius of the grid blocks containing the well connections. The productivity index is equal ot the sum over the connections of the product of the connection factor and the local mobility of the preferred phase.

If the value is negative, the well's potential flow rate is printed in the well reports instead of the productivity index. (The well's potential rate is the flow rate it would achieve in the absence of any rate constraint, limited only by its BHP and THP constraint),

9. flag of a special inflow equation to model the flow of gas between the completed grid blocks and the well completions: STD (or NO) – the standart inflow equations will be used, GPP – generalized pseudo-pressure equation (this option may be used by gas condensate producers – the keyword PICOND (see 6.10.64)). Generalized pseudo-pressure equation alters both the gas and oil mobilities and takes accounts also of the effects of condensate dropout,

10. well can be automatically shut or stopped if a well or group economic limit or group flow limit is violated: CodeSTOP - stop well above formation, SHUT - isolate well from the formation,

11. crossflow ability flag: YES - crossflow allowed in the well, NO - crossflow not allowed in the well, connection is a one-way valve, which prevents reverse drawdown flow.

Default:

- well group name: FIELD,

- reference depth for bottom hole pressure - first perforated interval depth,

- drainage radius – 0,

- flag of a special inflow equation – STD,

- automatically shut or stopped if a well or group economic limit or group flow limit is violated – SHUT,

- crossflow ability flag – YES.

```
Example
WELSPECL
WELL1 G1 LGR1 6 8 5500 /
WELL2 G1 LGR2 28 16 6000 /
/
```

In this example two wells are defined: WELL1, it's bottom hole situated at $X = 6$ and $Y = 8$, from group G1. This well is located in local grid refinement LGR1, reference depth – 5500м. WELL2, it's bottom hole situated at $X = 28$ and $Y = 16$, frm group G1. This well is located in local grid refinement LGR2, reference depth – 6000м.

6.10.4 WELSEGS

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

Nhe keyword specifies the segment structure of a multisegment well (section 2.15.2). One keyword can specify only one well. The well should be previously defined via the keyword WELSPECS (see 6.10.2) (WELSPECL (see 6.10.3)).

The keyword consists of the following data: first data record defines the well and its top segment (nearest to the wellhead), next data records defines other segments. Each data record should be terminated with a slash /. All data should be terminated with a final /.

First data record – the following data for the first segment should be entered:

1. well name (or well number);

2. depth of the nodal point of the top segment (BHP reference depth) (m);

3. length down the tubing to the nodal point of the top segment (m). The position of top segment's node is fixed relative to the zero tubing length reference point (from which all the tubing length values are measured). For this segment (above the nodal point of the top segment pressure losses are not calculated). Pressure losses between the BHP reference depth and the tubing head should normally be handled by VFP tables;

4. effective wellbore volume of the top segment (m^3);

5. type of tubing length and depth information entered in second (etc.) data records of this keyword:
   INC – length and depth values entered in parameter 5 and 6 of the next data records refer to the incremental changes of these quantities along each segment (increase gradually).
   ABS – length and depth values entered in parameter 5 and 6 of the next data records refer to the absolute values of the tubing length and depth at the segment nodes.

6. components of the pressure drop that will be included in the calculation for well's segments:
   HF- – hydrostatic + friction (tNavigator supports only these components);

7. default Multi-phase Flow Model for the well's segments:
   HO – homogeneous flow (the phases all flow with the same velocity) (tNavigator supports only this flow type);

Default:

- length down the tubing to the nodal point of the top segment – 0 (m).

- effective wellbore volume of the top segment – 1.0e-5 (m^3).

Second and other data records – the following data for the second segment should be entered (other segments – the same way):

1. segment number at the start of the range (nearest the top segment) (all parameters of this keyword (from 3) specify the properties od the segments in the range given by parameters 1 and 2. Parameter 1 should be equal to parameter 2 if one need to enter properties of one segment);

2. segment number at the end of the range (all parameters of this keyword (from 3) specify the properties od the segments in the range given by parameters 1 and 2. Parameter 1 should be equal to parameter 2 if one need to enter properties of one segment);

3. branch number (1 – segments situated on the main stem. Lateral branches should have numbers – 2 etc.);

4. number of outlet segment (number of the segment to which the segment at the start of the range (parameter 1) is joined, that is its neighbor in the direction of the wellhead). A node in a multi-lateral well can be the outlet segment of two or more segments of different branches;

5.   - if parameter 5 of first data record (for top segment) – INC:
       length of each segment in the range, that is the distance between its nodal point and that of its neighbor in the direction of the wellhead (m).

      - if parameter 5 of first data record (for top segment) – ABS:
       length down the tubing (from the zero tubing length reference point) at the nodal point of the last segment in the range. The length between this point and the node of the range's outlet segment (parameter 4) is divided into equal lengths (this value is equal to the number of segments in the range) (m).

6.   - if parameter 5 of first data record (for top segment) – INC:
       depth change along each segment in the range, that is the depth of its nodal point minus the depth of its outlet segment's nodal point (m).

      - if parameter 5 of first data record (for top segment) – ABS:
       depth of the nodal point of the last segment in the range. The depths of any other segments in this range is calculated by interpolating between this depth and the depth of the range's outlet segment (parameter 4), linearly according to their length down the tubing (m).

7. tubing internal diameter (m) (IGNORED) this is an Eclipse compatibility field;

8. effective absolute roughness of the tubing (m) (IGNORED) this is an Eclipse compatibility field;

9. cross-sectional area for fluid flow (m^2) (IGNORED) this is an Eclipse compatibility field;

10. segment's volume (m^3) (IGNORED) this is an Eclipse compatibility field;

11. 　● if parameter 5 of first data record (for top segment) – INC:
    length of each segment in the range projected onto the X-axis, that is the X-coordinate of its nodal point minus that of its neighbor in the direction of the wellhead (m).

　● if parameter 5 of first data record (for top segment) – ABS:
    X-coordinate of the nodal point of the last segment in the range, relative to the origin of the grid. X-coordinates of any other segments in this range is calculated by interpolating between this value and the X coordinate of the range's outlet segment (parameter 4), linearly according to their length down the tubing (m).

12. 　● if parameter 5 of first data record (for top segment) – INC:
    length of each segment in the range projected onto the Y-axis, that is the Y-coordinate of its nodal point minus that of its neighbor in the direction of the wellhead (m).

　● if parameter 5 of first data record (for top segment) – ABS:
    Y-coordinate of the nodal point of the last segment in the range, relative to the origin of the grid. Y-coordinates of any other segments in this range is calculated by interpolating between this value and the Y-coordinate of the range's outlet segment (parameter 4), linearly according to their length down the tubing (m).

Example
```
WELSEGS
'PROD' 7010 10 0.31 'INC' 'HF-' /
2 12 1 1 20 20 0.2 1.E-3 1* 1* /
13 13 2 2 50 10 0.2 1.E-3 1* 1* /
14 17 2 13 100 10 0.2 1.E-3 1* 1* /
18 18 3 9 50 10 0.2 1.E-3 1* 1* /
19 22 3 18 100 10 0.2 1.E-3 1* 1* /
23 23 4 12 50 10 0.2 1.E-3 1* 1* /
24 27 4 23 100 10 0.2 1.E-3 1* 1* /
/
```

In this example for the well PROD depth of the nodal point of the top segment – 7010 m, length down the tubing to the nodal point of the top segment – 10 m, effective wellbore volume of the top segment – 0.31 m^3, type of tubing length and depth information – INC, components of the pressure drop – hydrostatic + friction. There are 4 branches. First branch contains segments: 2-12. 2-nd branch: segments 13-17, 3-rd: segments 18-22, 4-th: segments 23-27.

6.10.5 WSEGTABL

	tNavigator		eclipse 300		MORE Roxar
√		√		☐	
√	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword defines calculation of segment pressure drop from VFP tables (section 2.15.2). Multi-segmebt well should be entered via the keyword WELSEGS (see 6.10.4), tables should be entered via VFPPROD (see 6.10.19).

The data for each well should be terminated with a slash /. All data should be terminated with a final slash /.

The following parameters should be specified:

1. well name (or number),

2. segment number at the start of the range (all parameters of this keyword (from 4) specify the properties od the segments in the range given by parameters 2 and 3. Parameter 2 should be equal to parameter 3 if one need to enter properties of one segment);

3. segment number at the end of the range (all parameters of this keyword (from 4) specify the properties od the segments in the range given by parameters 2 and 3. Parameter 2 should be equal to parameter 3 if one need to enter properties of one segment);

4. production VFP table number. Second parameter of the main data in the keyword VFPPROD (see 6.10.19) is either datum length or datum depth of the segment. Pressure drop along each segment in proportion to its length or depth span. The table may extend into negative flows, which are interpreted as flow in the direction away from the wellhead.

5. components of the pressure drop that are included in the VFP table. tNavigator supports only F- option – friction option. The hydrostatic pressure drop is calculated from the fluid density in the segment and added to the friction pressure drop interpolated from the VFP table.

Default:

- production VFP table number – 0. Pressure drop is calculated according gomogeneous flow model.

```
Example
WSEGTABL
WELL1 6 10 3 F- /
```

In this example for the well WELL1 for segments from 6 to 10 3-rd VFP table is specified.

6.10.6 COMPDAT

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines well completions. A set of data for every well should be terminated with a slash /.

The following parameters are to be specified:

1. well name (or number),

2. perforated interval coordinate in X direction (IW); there is i in 5.3.1,

3. perforated interval coordinate in Y direction (JW); there is j in 5.3.1,

4. number of layer where this vertical perforated interval starts (layers are numbered top-down starting from 1); this is number k in 5.3.1,

5. number of layer where this vertical perforated interval ends, this is number k in 5.3.1,

6. completion status (OPEN, SHUT or PATH — part of well trajectory. If well trajectory is specified using PATH then behind-the-casing-flow could be calculated – see the keywords WNNC (see 6.10.39), WCONNNC (see 6.10.40)),

7. saturation table number (IGNORED, this is an Eclipse compatibility field),

8. transmissibility factor for each connection in this interval (if it is specified, well diameter, skin and KH are ignored), this is coefficient $T(t)$ in 5.3.1,

9. well diameter, value d_w in 5.3.2,

10. effective KH (permeability x thickness) for each connection in this interval, (see 5.3.2),

11. skin factor, this is value s in 5.3.2,

12. well's D-factor for non-Darcy flow handling (the connection's D-factor can be entered here directly: one should specify it here with negative sign. tNavigator takes this value without negative sign). Well's D-factor may be entered via WDFAC (see 6.10.59);

13. direction in which this interval penetrates grid block: X, Y or Z,

14. effective radius, this is value r_o in 5.3.2.

This keyword may be defined several times at any time step for the same well for different connections. Perforated intervals for the same well may be spaced (may not form a continuous interval) and have different orientation in space.

Default:

1. non specified,

2. i is taken from 6.10.2,

3. j is taken from 6.10.2,

4. starting block of vertical perforated interval: 1,

5. ending block of vertical perforated interval: NZ,

6. connection status: OPEN,

7. non specified,

8. transmissibility factor: calculated,

9. well diameter: $d_w = 0.156$ m,

10. KH value: calculated,

11. skin factor: $s = 0$,

12. non specified,

13. well orientation in space: Z,

14. effective radius: calculated.

If all connections for a well are in SHUT mode then this well automatically will be switched to working mode SHUT (WCONPROD (see 6.10.12), WCONINJE (see 6.10.14)).

```
Example
DIMENS
25 20 3
...
COMPDAT
1043  14 10 1 2 OPEN 5* /
W13   16 13 1 3 OPEN 2* 0.16 20 1 1* Z 1* /
PRO4   5  5 1 1 OPEN 2* 0.16 3* X /
PRO4   5  5 2 2 OPEN 2* 0.16 1* -1 3* /
PRO4   5  5 3 3 OPEN 2* 0.16 50 2* Y /
/
```

This example defines perforation intervals for three wells, for a grid 25x20x3. All wells belong to the same group FIELD, perforations are open.

Well 1043 has coordinates $X = 14$, $Y = 10$, first two layers are perforated, well diameter, skin factor and orientation are taken by default: diameter $d_w = 0.156$ m, skin $s = 0$ and

orientation in Z direction; KH and effective radius are calculated.

Well W13 has coordinates $X = 16$, $Y = 13$, layers from the first one to the third one are perforated, diameter is set to $d_w = 0.16$ m, skin is $s = 1$, KH = 20 and orientation in space is in Z direction.

The third well, PRO4, is also perforated in all three layers: in the first layer skin and KH are defaulted, orientation in space is in X direction; in the second layer skin is equal to $s = -1$, KH and orientation in space are defaulted (orientation=Z), in the third layer default skin factor is taken $s = 0$, KH multiplier is set to KH = 50 and orientation in space is in Y direction.

6.10.7   COMPDATL

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines well completions in local grids – section 2.21. A set of data for every well should be terminated with a slash /.

COMPDATL must be used in place of COMPDAT (see 6.10.6) to specify the connection data for wells in local refined grids, after the wells have been introduced with keyword WELSPECL (see 6.10.3) (instead of WELSPECS (see 6.10.2)). The keyword data for COMPDATL (see 6.10.7) is the same as for COMPDAT (see 6.10.6), except for an extra parameter 2 which names the local grid containing the connections specified in the record.

The following parameters are to be specified:

1. well name (or number),

2. local grid name (CARFIN (see 6.2.61)),

3. perforated interval coordinate in X direction (IW); there is i in 5.3.1,

4. perforated interval coordinate in Y direction (JW); there is j in 5.3.1,

5. number of layer where this vertical perforated interval starts (layers are numbered top-down starting from 1); this is number k in 5.3.1,

6. number of layer where this vertical perforated interval ends, this is number k in 5.3.1,

7. completion status (OPEN, SHUT or PATH — part of well trajectory. If well trajectory is specified using PATH then behind-the-casing-flow could be calculated – see the keywords WNNC (see 6.10.39), WCONNNC (see 6.10.40)),

8. saturation table number (IGNORED, this is an Eclipse compatibility field),

9. transmissibility factor for each connection in this interval (if it is specified, well diameter, skin and KH are ignored), this is coefficient $T(t)$ in 5.3.1,

10. well diameter, value d_w in 5.3.2,

11. effective KH (permeability x thickness) for each connection in this interval, (see 5.3.2),

12. skin factor, this is value s in 5.3.2,

13. well's D-factor for non-Darcy flow handling (the connection's D-factor can be entered here directly: one should specify it here with negative sign. tNavigator takes this value without negative sign). Well's D-factor may be entered via WDFAC (see 6.10.59);

14. direction in which this interval penetrates grid block: X, Y or Z,

15. effective radius, this is value r_o in 5.3.2.

This keyword may be defined several times at any time step for the same well for different connections. Perforated intervals for the same well may be spaced (may not form a continuous interval) and have different orientation in space.

   Default:

1. non specified,

2. non specified,

3. i is taken from 6.10.2,

4. j is taken from 6.10.2,

5. starting block of vertical perforated interval: 1,

6. ending block of vertical perforated interval: NZ,

7. connection status: OPEN,

8. non specified,

9. transmissibility factor: calculated,

10. well diameter: $d_w = 0.156$ m,

11. KH value: calculated,

12. skin factor: $s = 0$,

13. non specified,

14. well orientation in space: Z,

15. effective radius: calculated.

```
Example
DIMENS
25 20 3
...
COMPDAT
1043 LGR1 14 10 1 2 OPEN 5* /
W13  LGR2 16 13 1 3 OPEN 2* 0.16 20 1 1* Z 1* /
PRO4    5  5 1 1 OPEN 2* 0.16 3* X /
PRO4    5  5 2 2 OPEN 2* 0.16 1* -1 3* /
PRO4    5  5 3 3 OPEN 2* 0.16 50 2* Y /
/
```

This example defines perforation intervals for three wells, for a grid 25x20x3. All wells belong to the same group FIELD, perforations are open.

Well 1043 has coordinates $X = 14$, $Y = 10$, first two layers are perforated, well diameter, skin factor and orientation are taken by default: diameter $d_w = 0.156$ m, skin $s = 0$ and orientation in Z direction; KH and effective radius are calculated. This well is located in local grid refinement LGR1.

Well W13 has coordinates $X = 16$, $Y = 13$, layers from the first one to the third one are perforated, diameter is set to $d_w = 0.16$ m, skin is $s = 1$, KH = 20 and orientation in space is in Z direction. This well is located in local grid refinement LGR2.

The third well, PRO4, is also perforated in all three layers: in the first layer skin and KH are defaulted, orientation in space is in X direction; in the second layer skin is equal to $s = -1$, KH and orientation in space are defaulted (orientation=Z), in the third layer default skin factor is taken $s = 0$, KH multiplier is set to KH = 50 and orientation in space is in Y direction.

6.10.8 COMPLUMP

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword lumps connections together into completions to provide realization of simultaneous actions. Completions numbers could be used in the keywords WPIMULT (see 6.10.10), WBHZONE (see 6.10.50), WPIFUNC (see 6.10.48), WSKFUNC (see 6.10.49). Connections should first have been defined with the keyword COMPDAT (see 6.10.6). When a connection is first defined with COMPDAT (see 6.10.6), it is given a completion number equal to its order of introduction in the well. If the keyword COMPLUMP (see 6.10.8) is not used, each connection is in its own separate completion and the terms completion and connection are equivalent.

Any number of data records could be specified. Each record should be terminated with a slash /. All data should be terminated with a final slash /.
Each record should contain the following parameters:

1. well name (or number);

2. connection coordinate in X direction;

3. connection coordinate in Y direction;

4. coordinate of upper connection block in this completion;

5. coordinate of lower connection block in this completion;

6. completion number of these connections.

Default:

- connection coordinate in X direction – 0 (any value);

- connection coordinate in Y direction – 0 (any value);

- coordinate of upper connection block in this completion – 0 (the top connection of this well);

- coordinate of lower connection block in this completion – 0 (the bottom connection of this well);

The set of connections in the well which match the location indices specified in parameters 2-5 will be given the completion number specified in parameter 6. All connections having the same number of completion are treated as belonging to the same completion and can be used together in the keywords WPIMULT (see 6.10.10), WBHZONE (see 6.10.50), WPIFUNC (see 6.10.48), WSKFUNC (see 6.10.49).
If a location index is defaulted to zero, it plays no part in selecting the set of connections.

Thus if X and Y coordinates are defaulted to zero, all connections in the well that are located between layers specified in parameters 4 and 5 will be given the completion number specified in parameter 6. If all four location indices are defaulted, all the well's connections will be given this completion number.

Example
```
COMPLUMP
PROD 2* 1 2 1 /
PROD 2* 3 4 2 /
PROD 2* 5 5 3 /
/
```

In this example for well **PROD**: connections in layers 1 and 2 have the completion number 1, connections in layers 3 and 4 have the completion number 2, connections in layer 5 have the completion number 3.

6.10.9   COMPLMPL

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword lumps connections together into completions to provide realization of simultaneous actions for the wells in local grids (section 2.21). Completions numbers could be used in the keywords WPIMULT (see 6.10.10). Connections should first have been defined with the keyword COMPDATL (see 6.10.7). When a connection is first defined with COMPDATL (see 6.10.7), it is given a completion number equal to its order of introduction in the well. If the keyword COMPLMPL (see 6.10.9) is not used, each connection is in its own separate completion and the terms completion and connection are equivalent.

COMPLMPL must be used in place of COMPLUMP (see 6.10.8) to lumps connections together into completions for wells in local refined grids, after the wells have been introduced with keywords WELSPECL (see 6.10.3), COMPDATL (see 6.10.7). The keyword data for COMPLMPL is the same as for COMPLUMP (see 6.10.8), except for an extra parameter 2 which names the local grid containing the connections specified in the record.

Any number of data records could be specified. Each record should be terminated with a slash /. All data should be terminated with a final slash /.

Each record should contain the following parameters:

1. well name (or number);

2. local grid name (CARFIN (see 6.2.61));

3. connection coordinate in X direction;

4. connection coordinate in Y direction;

5. coordinate of upper connection block in this completion;

6. coordinate of lower connection block in this completion;

7. completion number of these connections.

Default:

- connection coordinate in X direction – 0 (any value);

- connection coordinate in Y direction – 0 (any value);

- coordinate of upper connection block in this completion – 0 (the top connection of this well);

- coordinate of lower connection block in this completion – 0 (the bottom connection of this well);

The set of connections in the well which match the location indices specified in parameters 3-6 will be given the completion number specified in parameter 7. All connections having the same number of completion are treated as belonging to the same completion and can be used together in the keywords WPIMULT (see 6.10.10).

If a location index is defaulted to zero, it plays no part in selecting the set of connections. Thus if X and Y coordinates are defaulted to zero, all connections in the well that are located between layers specified in parameters 5 and 6 will be given the completion number specified in parameter 7. If all four location indices are defaulted, all the well's connections will be given this completion number.

```
Example
COMPLUMP
PROD 'LGR3' 2* 1 2 1 /
PROD 'LGR3' 2* 3 4 2 /
PROD 'LGR3' 2* 5 5 3 /
/
```

In this example for well PROD (situated in local grid LGR3): connections in layers 1 and 2 have the completion number 1, connections in layers 3 and 4 have the completion number 2, connections in layer 5 have the completion number 3.

6.10.10 WPIMULT

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword is used to multiply well connection transmissibility factors by specified value. A set of data for every well should be terminated with a slash **/**. The following parameters are to be specified:

1. well name (or number),

2. connection transmissibility factor multiplier,

3. perforated interval coordinate in X direction, this is i in 5.3.1,

4. perforated interval coordinate in Y direction, this is j in 5.3.1,

5. perforated interval coordinate in Z direction, this is k in 5.3.1.

6. number of first completion in range (completion numbers are set via COMPLUMP (see 6.10.8)),

7. number of last completion in range (completion numbers are set via COMPLUMP (see 6.10.8)).

If WPIMULT is used several times for the same connection transmissibility factor (such specifications of WPIMULT should be separated by keywords DATES (see 6.10.33) or TSTEP (see 6.10.34)), its effect is cumulative, i.e. multiplying factor is applied each time the keyword is entered.

If a COMPDAT (see 6.10.6) keyword contains data inducing transmissibility factor recalculation, multiplier previously defined by WPIMULT is reset to 1.0.

Default:

- connection transmissibility factor multiplier — 1.0,

- coordinate in X direction — any,

- coordinate in Y direction — any,

- coordinate in Z direction — any,

- number of first completion in range — any,

- number of last completion in range — any.

```
Example
DIMENS
25 20 3
...
DATES
1 JUN 2003
/
COMPDAT
1043 14 10 1 2 OPEN 1* 5 /
1044 20 10 1 1 OPEN 8* /
/
...
DATES
1 JUN 2004
/
WPIMULT
1043 2 14 10 1 /
1044 2 20 10 1 /
/
...
DATES
1 JUN 2005
/
WPIMULT
1043 0.25 14 10 1 /
1044 2 /
1020 2 1 /
/
```

In this example on the 1st of June, 2003 new completions are specified for wells 1043 and 1044. For well 1043 completion factor is specified explicitly and is equal to 5. In a year completion factors for both wells in the first layer are multiplied by 2; so, completion factor for well 1043 is now equal to 10.

In one more year it is multiplied again on 0.25 and becomes equal to 2.5. At the same time completion factors for all completions of well 1044 are multiplied by 2, and for well 1020 only those with i-coordinate equal to 1 are doubled.

6.10.11   WPIMULTL

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword is used to multiply well connection transmissibility factors by specified value for the wells in local grids (section 2.21). A set of data for every well should be terminated with a slash /.

WPIMULTL must be used in place of WPIMULT (see 6.10.10) to multiply well connection transmissibility factors by specified value for wells in local refined grids, after the wells have been introduced with keywords WELSPECL (see 6.10.3), COMPDATL (see 6.10.7). The keyword data for WPIMULT is the same as for WPIMULT (see 6.10.10), except for an extra parameter 3 which names the local grid containing the connections specified in the record.

The following parameters are to be specified:

1. well name (or number),

2. connection transmissibility factor multiplier,

3. local grid name (CARFIN (see 6.2.61));

4. perforated interval coordinate in X direction, this is i in 5.3.1,

5. perforated interval coordinate in Y direction, this is j in 5.3.1,

6. perforated interval coordinate in Z direction, this is k in 5.3.1.

7. number of first completion in range (completion numbers are set via COMPLMPL (see 6.10.9)),

8. number of last completion in range (completion numbers are set via COMPLMPL (see 6.10.9)).

If WPIMULTL is used several times for the same connection transmissibility factor (such specifications of WPIMULTL should be separated by keywords DATES (see 6.10.33) or TSTEP (see 6.10.34)), its effect is cumulative, i.e. multiplying factor is applied each time the keyword is entered.

If a COMPDATL (see 6.10.7) keyword contains data inducing transmissibility factor recalculation, multiplier previously defined by WPIMULTL is reset to 1.0.

Default:

- connection transmissibility factor multiplier — 1.0,

- coordinate in X direction — any,

- coordinate in Y direction — any,

- coordinate in Z direction — any,

- number of first completion in range — any,

- number of last completion in range — any.

```
Example
DIMENS
25 20 3
...
DATES
1 JUN 2003
/
COMPDATL
1043 'LGR1' 14 10 1 2 OPEN 1* 5 /
1044 'LGR1' 20 10 1 1 OPEN 8* /
/
...
DATES
1 JUN 2004
/
WPIMULTL
1043 'LGR1' 2 14 10 1 /
1044 'LGR1' 2 20 10 1 /
/
...
DATES
1 JUN 2005
/
WPIMULTL
1043 'LGR1' 0.25 14 10 1 /
1044 'LGR1' 2 /
1020 'LGR1' 2 1 /
/
```

In this example on the 1st of June, 2003 new completions are specified for wells 1043 and 1044 (situated in local grid LGR1). For well 1043 completion factor is specified explicitly and is equal to 5. In a year completion factors for both wells in the first layer are multiplied by 2; so, completion factor for well 1043 is now equal to 10.

In one more year it is multiplied again on 0.25 and becomes equal to 2.5. At the same time completion factors for all completions of well 1044 are multiplied by 2, and for well 1020 only those with i-coordinate equal to 1 are doubled.

6.10.12 WCONPROD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword could be specified for a well or a group of wells that work as producing. A set of data for every well should be terminated with a slash /.

The following parameters should be specified:

1. well name (or number),

2. well working mode (OPEN, STOP, SHUT or AUTO - well connection will open automatically when the value of oil saturation runs up to WELSOMIN (see 6.10.1)),

3. well control (LRAT — liquid rate control, ORAT — oil rate control, WRAT — water rate control, GRAT — gas rate control, RESV — reservoir liquid rate control, THP — tubing head pressure control or BHP — bottom hole pressure control); during history match mode it indicates which parameter is used for rate calculation,

4. oil rate (or limit),

5. water rate (or limit),

6. gas rate (or limit),

7. liquid rate on the surface (or limit),

8. liquid rate in reservoir conditions (or limit),

9. BHP value or limit,

10. well VFP table number (VFPPROD (see 6.10.19)); if zero, THP will not be reported.

User should specify well control and, if needed, rate and pressure limits. If limit is not set or set to zero, default value will be used. Well is first set to control specified by user. Then all limits are checked, if one of them is violated, well will switch to corresponding control. Limits check will proceed until all of them are satisfied. If well produces negative rate according to its current control, it is closed. Otherwise simulator will make step with this control. Next step the procedure is repeated. Bottom hole pressure p_{BH} for rate control wells and rates for pressure control wells are calculated from (2.62).

All well control switching times are logged, , (see 6.1.2) section WELL.

Default:

- well status: OPEN,

- liquid rate limit: $1e+20$ m^3 / day,

- BHP limit: p_{atm} (atmosphere pressure).

Example
```
WCONPROD
1043 OPEN LRAT 18.19      0 0 18.99 2* /
1054 OPEN ORAT 16.38 1.765 0 18.14 1* 50 /
/
```

This example sets controls for two wells. Well 1043 is on liquid rate control; total liquid rate in surface conditions is 18.99. There is no bottom hole pressure limit, so the simulator will set liquid production equal to 18.99; if calculated pressure is lower than atmospheric (default bottom hole limit is atmospheric pressure), then the well will switch to bottom hole pressure control. If calculated oil rate is above 18.19, well will switch to oil rate control.

Well 1054 is on oil rate 16.38 control. There is a BHP limit for this well, equal to 50; so, if pressure goes below 50, the well will automatically switch to BHP control. Liquid and water controls are also checked each time; if one of them is violated, well is switched to corresponding control.

Example
```
WCONPROD
43 OPEN BHP 18.19      0 0 18.9 1* 55 /
54 OPEN BHP 16.38 1.765 0 1*    1* 45 /
/
```

In this example both wells are on BHP control. In this case rates are calculated by simulator. If liquid rate for well 43 exceeds 18.9, it will switch to LRAT control. If oil rate exceeds 18.19, well will switch to ORAT control.

Correspondingly, if water rate for well 54 exceeds 1.765 (or oil rate exceeds 16.38), it will switch to WRAT control (correspondingly, ORAT control).

6.10.13 WCONHIST

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword may be specified for a well or a group of wells (instead of the keyword WCONPROD (see 6.10.12)) that work as producers and are being history matched. The keyword WCONINJH (see 6.10.16) is used for wells that work as injectors and are being history matched.

A set of data for every well should be terminated with a slash /. The following data should be specified:

1. well name (or number),

2. well status (OPEN, STOP or SHUT),

3. well control (LRAT — liquid rate control, ORAT — oil rate control, WRAT — water rate control, GRAT — gas rate control, RESV — reservoir liquid rate control);

4. observed oil rate,

5. observed water rate,

6. observed gas rate,

7. well VFP table number (VFPPROD (see 6.10.19)); if zero, THP will not be reported; if non-zero, THP will be calculated and reported,

8. IGNORED, this is an Eclipse compatibility field,

9. observed tubing head pressure THP, will be used for reporting, it's not a limit,

10. observed bottom hole pressure BHP, will be used for reporting, it's not a limit.

Well defined by this keyword is on rate control. Control value will be calculated on basis of historical data; in case of LRAT limit rate is calculated as sum of historical water and oil rates. Bottom hole pressure (and tubing head pressure) are indicated for historical purposes and these are not limits.

To limit bottom hole pressure, use WELTARG keyword, see 6.10.17. Otherwise default BHP limit equal to atmospheric pressure p_{atm} is used.

If calculated p_{BH} is lower than BHP limit, the well will switch to BHP control. During the calculation process the simulator checks whether the well can switch back to specified rate control. If the switch doesn't violate BHP limit, the well will return to rate control mode.

All well control switching times are logged, , (see 6.1.2) section WELL.

This keyword is convenient for history matching purposes.

Default:

- well status: OPEN,

- well VFP table number (VFPPROD (see 6.10.19)): last indicated value

- observed tubing head pressure, THP: 0

- observed bottom hole pressure, BHP: 0

- bottom hole pressure limit: p_{atm} (atmospheric pressure)

```
Example
WCONHIST
1043 OPEN LRAT 18.9        0 0 /
1054 OPEN LRAT 16.38 1.765 0 3* 57 /
/
```

This example sets controls for two wells. Well 1043 is on liquid rate control; oil rate is equal to 18.9, water and gas are not produced, hence total liquid rate is 18.9. The simulator will set liquid production equal to 18.9; if calculated pressure is lower than atmospheric (default bottom hole limit is atmospheric pressure), then the well will switch to bottom hole pressure control.

Well 1054 is also on liquid rate control; oil and water rates are equal to 16.38 and 1.765; hence total surface liquid rate is 18.145. There is a historical BHP value equal to 57.

6.10.14   WCONINJE

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword could be specified for a well or a group of wells that work as injectors. A set of data for every well should be terminated with a slash /. The following data should be specified:

1. name of well (or number),

2. injected fluid (WATER (WAT is the same), OIL or GAS),

3. well working mode (OPEN, STOP or SHUT),

4. well control (RATE — controlled by surface flow rate target, BHP — controlled by BHP target, THP — controlled by THP target, RESV — controlled by reservoir volume rate target, GRUP — group control (the keyword GCONINJE (see 6.10.24))),

5. surface rate of injected fluid (or limit),

6. reservoir rate of injected fluid,

7. BHP for injection or limit,

8. THP for injection or limit.

User may specify one of the following combinations:

- Specify BHP value p_{BH}, and set well to BHP control, in this case injection rate is calculated from (2.62). If injection rate exceeds limit, the well will switch to RATE control.

  During the calculation the simulator will check whether the rate limit is still violated if the well switches to BHP control. If not, the well will switch back.

- Set the well to RATE control, and specify well injection rate; in this case if computed BHP is higher than the BHP limit, then well control is switched to BHP with BHP value being equal to this limit.

  During the calculation the simulator will check whether the BHP limit is still violated if the well switches to RATE control. If not, the well will switch back.

All well control switching times are logged, , (see 6.1.2) section WELL.
  Default:

- well status: OPEN,

- injection rate limit: 1000000 m^3 / day

• BHP limit: $6801 p_{atm}$

```
Example
WCONINJE
1043 WATER OPEN RATE 253.1 2* /
1054 WATER OPEN RATE 253.1 1* 400/
/
```

This example sets controls for two wells. Well 1043 is on injection rate control, the rate is equal to 253.1. There is no bottom hole pressure limit, so the simulator will set injection rate equal to 253.1. If the user specifies an injection rate so high that necessary bottom hole pressure exceeds default limit ($6801 p_{atm}$), the well will automatically switch to BHP control.

Well 1054 is also on injection rate control, the rate is equal to 253.1. This well has bottom hole pressure limit equal to 400; so, if calculated pressure goes above 400, the well will automatically switch to BHP control.

```
Example
WCONINJE
43 WATER OPEN BHP 255 1* 55/
54 WATER OPEN BHP 255 1* 45/
/
```

In this example both wells are on BHP control. In this case injection rate is calculated by the simulator. If it exceeds 255, well will switch to RATE control.

6.10.15 WCONINJP

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the injector which injects at a rate equal to the sum of parts of volumes of production (reservoir volume) of neighboring producers. For example, in a five spot pattern one can specify the injector whose injection rate is equal to the sum of one quaters of the production volume of its four neighboring producers.

The keyword can specify this type of control only for one well. If there are several injectors with this type of control, one should use this keyword several times.

If the well has this type of control, it can't have the control of group it belongs to. So this well can switch only from this type of control to BHP limit and vice versa.

The data consists of two parts: 1st part – description of injector (this part should be terminated with a slash /), 2nd part – producers, which influence on the injection volume of the injector in part 1 (the data of each producer should be terminated with a slash /). The 2nd part should be terminated with single slash /.

The following data should be specified (Part 1):

1. name of well (or number),

2. injected fluid (WATER (WAT is the same), OIL or GAS),

3. well working mode (OPEN, STOP or SHUT),

4. BHP for injection or limit,

5. IGNORED, this is an Eclipse compatibility field,

6. IGNORED, this is an Eclipse compatibility field,

7. IGNORED, this is an Eclipse compatibility field,

8. factor; calculated injection volume for this injector will be multiplied by this factor.

Default:

* injected fluid – WATER,

* well working mode – OPEN,

* factor – 1.

Part 2. The following data should be specified for each well:

1. producer name (or number)

2. the fraction (a value between 0 and 1). This fraction is equal to the part of production volume of this producer. Injector in part 1 will inject at a rate equal to the sum of parts of volumes of production of corresponding producers. **Default**. This value is re-computing at each time step if an injector (corresponding to this producer) with the same type of control starts working or stops.

3. fluid-in-place region number.

```
Example
WCONINJP
WELL1 WATER OPEN 3500/
P1 1 /
P2 0.25 /
P3 0.25 /
P4 0.5 /
/
```

This example sets the injector WELL1 is open, injects water and has BHP limit 3500. WELL1 injects at a rate equal to the sum of following volumes: total production volume of P1, one quater of production volume of P2, one quater of production volume of P3 and one half of production volume of P4.

6.10.16 WCONINJH

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword may be specified for a well or a group of wells (instead of the keyword WCONINJE (see 6.10.14)) that work as injectors and are being history matched. The keyword WCONHIST (see 6.10.13) is used for wells that work as producers and are being history matched.

A set of data for every well should be terminated with a slash /.

The following data should be specified:

1. name of well (or number),

2. injected fluid (WATER (WAT is the same), OIL or GAS),

3. well working mode (OPEN, STOP or SHUT),

4. observed surface rate of injected fluid,

5. observed bottom hole pressure BHP, will be used for reporting, it's not a limit (well will not switch to BHP control if the calculated BHP is above this value),

6. observed tubing head pressure THP, will be used for reporting, it's not a limit (well will not switch to BHP control if the calculated BHP is above this value),

7. well VFP table number,

8. oil concentration in the injected gas or gas concentration in the injected oil, IGNORED, this is an Eclipse compatibility field,

Well defined by this keyword is on injection rate control. Bottom hole pressure (and tubing head pressure) are indicated for historical purposes and these are not limits.

To limit bottom hole pressure, use **WELTARG** keyword, see 6.10.17. Otherwise default BHP limit equal to atmospheric pressure p_{atm} is used.

If calculated p_{BH} is lower than BHP limit, the well will switch to BHP control. During the calculation process the simulator checks whether the well can switch back to inlection rate control. If the switch doesn't violate BHP limit, the well will return to injection rate control mode.

All well control switching times are logged, , (see 6.1.2) section WELL.

This keyword is convenient for history matching purposes.

Default:

- well status: OPEN,

- injector type – WATER,

- observed tubing head pressure, THP: 0

- observed bottom hole pressure, BHP: 0

- bottom hole pressure limit: p_{atm} (atmospheric pressure)

Example
```
WCONINJH
1043 WATER OPEN 2400 4* /
1054 GAS OPEN 3890 3620 2742 2 1*/
/
```

In this example there are two injectors that are being history matched. Well 1043 injects water, observed surface rate of injected fluid – 2400. Well 1054 injects gas, observed surface rate of injected fluid – 3890, observed BHP – 3620, observed THP – 2742, VFP table number - 2.

6.10.17 WELTARG

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword resets control or limit value for the well, defined by WCONPROD (see 6.10.12), WCONINJE (see 6.10.14), WCONHIST (see 6.10.13). The following options are available:

1. name of well (or number),

2. control or limit to be changed:

   - ORAT – oil rate
   - WRAT – water rate
   - GRAT – gas rate
   - LRAT – liquid rate
   - BHP – bottom hole pressure
   - THP – tubing head pressure
   - VFP – VFP table number (VFPPROD (see 6.10.19))

3. new value of this control or limit

Default: none
For a well on BHP control, WELTARG will simply reset control value.
For a well on rate control, WELTARG will change the BHP limit.

```
Example
WELTARG
245 BHP 20/
/
```

In this example bottom hole pressure limit (or control) is changed for well 245 using WELTARG.

```
Example
WCONHIST
p45 OPEN LRAT 18.9 0 0 3* 67 /
/
WELTARG
p45 BHP 30/
/
```

In this example first historical production and control mode are defined for well p45. Historical bottom hole pressure 67 is indicated. Then bottom hole pressure limit is set using WELTARG.

6.10.18 VFPINJ

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword inputs a table of BHP values versus FLO (oil, water or gas injection rate), THP (tubing head pressure) for injectors.

If there are wells defined that require THP calculation, at least one vertical flow table should be input. If several tables are used in calculation, each one is input separately by **VFPINJ** keyword.The table consists of the following records of data, each record terminated with a slash (/).

1. Basic data

   - table number (a number between 1 and the 3-rd parameter of **VFPIDIMS** (see 6.1.17));

   - bottom hole datum depth for this table;

   - definition of variable FLO (OIL – Oil injection rate, WAT – water injection rate, or GAS – Gas injection rate);

   - definition of variable THP (only THP value is supported);

   - units of this VFP table (METRIC or FIELD);

   - definition of the tabulated quantity in the body of the table, parameter number 4 (BHP – bottom hole pressure).

2. FLO (flow rate) values (NFLO values), entered in ascending order, terminated with a slash (/) (the number NFLO should not be greater than the 1-st parameter of **VFPIDIMS** (see 6.1.17)).

3. THP values (NTHP values), entered in ascending order, terminated with a slash (/) (the number NTHP should not be greater than the 2-nd parameter of **VFPIDIMS** (see 6.1.17)).

4. Next records, from fourth to NTHP+6.
   NTHP successive records each containing the following items, and each terminated with a slash (/).

   - NT (THP value number);

   - BHP at 1st FLO value, NTth THP value;

   - BHP at 2nd FLO value, NTth THP value;

   - ....

   - BHP at Last FLO value, NTth THP value.

All combinations of NF = 1 to NFLO, NT = 1 to NTHP must be covered.

Example
```
VFPINJ
1 5200 'WAT' 'THP' 'METRIC' 'BHP'/
1 200 800 900 1500 /
2500 /
1000 /
1 1.75243E+003 1.75243E+003 1.75244E+003 1.80749E+003
1.80749E+003
/
```

This example defines a VFP table for water injector: NFLO is equal to 5, NTHP − 1.

6.10.19   VFPPROD

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword inputs a table of BHP values versus FLO, versus THP, versus WFR, versus GFR, where

FLO is the oil, liquid or gas production rate

THP is the tubing head pressure

WFR is the water-oil ratio, water cut or water-gas ratio

GFR is the gas-oil ratio, gas-liquid ratio or oil-gas ratio.

If there are wells defined that require THP calculation, at least one vertical flow table should be input. If several tables are used in calculation, each one is input separately by **VFPPROD** keyword.The table consists of the following records of data, each record terminated with a slash (/).

1. Basic data

   - table number (a number between 1 and the 6-th parameter of **VFPPDIMS** (see 6.1.18));
   - bottom hole datum depth for this table;
   - definition of variable FLO (OIL – Oil production rate, LIQ – Liquid (oil + water) production rate, or GAS – Gas production rate);
   - definition of variable WFR (WOR – Water-oil ratio, WCT – Water cut (water-liquid ratio), or WGR – Water-gas ratio);
   - definition of variable GFR (GOR – Gas-oil ratio, GLR – Gas-liquid ratio, or OGR – Oil-gas ratio);
   - definition of variable THP (only THP value is supported);
   - IGNORED, This is an Eclipse compatibility field;
   - units of this VFP table (METRIC or FIELD);
   - definition of the tabulated quantity in the body of the table, parameter number 7 (BHP – bottom hole pressure).

2. FLO (flow rate) values (NFLO values), entered in ascending order, terminated with a slash (/) (the number NFLO should not be greater than the 1-st parameter of **VFPPDIMS** (see 6.1.18)).

3. THP values (NTHP values), entered in ascending order, terminated with a slash (/) (the number NTHP should not be greater than the 2-nd parameter of **VFPPDIMS** (see 6.1.18)).

4. WFR values (NWFR values), entered in ascending order, terminated with a slash (/) (the number NWFR should not be greater than the 3-rd parameter of **VFPPDIMS** (see 6.1.18)).

5. GFR values (NGFR values), entered in ascending order, terminated with a slash (/) (the number NGFR should not be greater than the 4-th parameter of **VFPPDIMS** (see 6.1.18)).

6. IGNORED, This is an Eclipse compatibility field.

7. Next records, from seventh to NTHP*NWFR*NGFR+6.
   NTHP*NWFR*NGFR successive records each containing the following items, and each terminated with a slash (/).

   - NT (THP value number);
   - NW (WFR value number);
   - NG (GFR value number);
   - IGNORED, This is an Eclipse compatibility field;
   - BHP at 1st FLO value, NTth THP value, NWth WFR value, NGth GFR value;
   - BHP at 2nd FLO value, NTth THP value, NWth WFR value, NGth GFR value;
   - ....
   - BHP at Last FLO value, NTth THP value, NWth WFR value, NGth GFR value.

All combinations of NF = 1 to NFLO, NT = 1 to NTHP, NW = 1 to NWFR, NG = 1 to NGFR must be covered.

Example
```
VFPPROD
1 2200 'OIL' 'WCT' 'GOR' 'THP' " 'METRIC' 'BHP'/
1 30 300 /
10 20 /
0 0.7 /
1 100 500 /
0 /
1 1 1 1 1.75243E+002 1.75243E+002 1.75244E+002 /
2 1 1 1 1.80749E+002 1.80749E+002 1.80750E+002 /
1 2 1 1 1.91358E+002 1.91359E+002 1.91362E+002 /
2 2 1 1 1.96743E+002 1.96744E+002 1.96747E+002 /
1 1 2 1 1.71599E+002 1.71599E+002 1.71601E+002 /
2 1 2 1 1.77093E+002 1.77093E+002 1.77095E+002 /
1 2 2 1 1.88482E+002 1.88483E+002 1.88487E+002 /
2 2 2 1 1.93865E+002 1.93866E+002 1.93869E+002 /
1 1 3 1 1.45582E+002 1.45526E+002 1.45462E+002 /
2 1 3 1 1.50977E+002 1.50978E+002 1.50979E+002 /
1 2 3 1 1.71277E+002 1.71278E+002 1.71282E+002 /
2 2 3 1 1.71277E+002 1.71278E+002 1.71282E+002 /
```

This example inputs table of BHP values for 3 different oil rate values, two different THP-s, two different water cuts and three gas oil ratios.

6.10.20   WECON

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines economic limit for production wells. The following parameters are to be specified:

1. name of well (or number),

2. lower economic limit of oil rate (in case of violation well will be shut),

3. IGNORED, this is an Eclipse compatibility field,

4. upper economic limit of water cut,

5. upper economic limit of gas-oil ratio,

6. upper economic limit of water-gas ratio,

7. workover action for watercut violation, gas-oil ratio violation or water-gas violation:

   - NONE – do nothing
   - CON – shut in the worst offending perforation,
   - CON+ – close worst offending perforation and all below,
   - +CON – the same as CON+,
   - WELL – shut in the well

Default:

- lower oil rate economic limit — 0,

- upper water cut economic limit — no limit,

- workover action: NONE.

```
Example
WECON
p45 3 1* 0.95 2* CON /
p35 3 4* WELL /
/
```

This example sets limits for two wells. For well p45 minimum possible oil rate is equal to 3, and maximum possible water cut is 95%. If oil rate falls below 3, well will be shut in; if watercut exceeds 95%, the worst offending perforation will be closed.

For well p35 minimum possible oil rate is also equal to 3, and there is no water cut limit. If calculated oil rate is lower than 3, the well will be automatically shut in.

6.10.21   WECONINJ

☑ **tNavigator** ☐ **eclipse 300** ☐ **MORE Roxar**	
☑ **eclipse 100** ☐ **CMG imex** ☐ **CMG stars**	

This keyword defines economic limit for injection wells. The data (for every well) should be terminated with a slash `/`. The following parameters are to be specified:

1. name of well (or number),

2. economic limit for water injection rate, if the injection rate falls below the specified minimum, the well is shut.

Default: none

```
Example
WECONINJ
i35 10/
/
```

This example sets minimum possible injection rate for well i45 equal to 10.

6.10.22   WEFAC

√ tNavigator	√ eclipse 300	☐ MORE Roxar
√ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword defines well efficiency factor. The following parameters are to be specified:

1. name of well (or number),

2. well efficiency factor,

3. IGNORED, this is an Eclipse compatibility field.

Default: 1

```
Example
WEFAC
P25 .89 /
I*  .97 /
/
```

In this example well P25 efficiency factor is set to 89%. For all wells with name starting from letter I efficiency factor is set to 97%.

6.10.23 GCONPROD

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword could be specified for a well group or several groups with the same group name root. The following parameters should be specified:

1. group name (or groop name root, i.e. name ending with asterisk), or FIELD (for field control),

2. control mode: NONE, no immediate control, LRAT, liquid rate control, ORAT, oil rate control, WRAT, water rate control, GRAT, gas rate control; it indicates which parameter is used for group rate calculation,

3. oil rate (or limit),

4. water rate (or limit),

5. gas rate (or limit),

6. liquid rate on the surface (or limit),

7. workover action for oil rate violation, water rate violation, gas rate violation or liquid rate violation:

   - NONE – do nothing
   - CON – shut in the worst offending perforation (in the worst offending well),
   - CON+ – close worst offending perforation and all below (in the worst offending well),
   - +CON – the same as CON+,
   - WELL – shut in the worst offending well.

8. group availability for higher group production rate target: YES means that group will switch to control from higher group if higher group level limit is violated; NO means that group rate is not influenced by controls from above groups,

9. groups's share of a higher rate of phase production (OIL, WAT, GAS or LIQ); phase is set using the next parameter of this keyword,

10. phase for the previous parameter of this keyword: OIL, WAT, GAS or LIQ.

If user specifies group control, and control mode is not NONE, simulator will modify rates of wells (or groups) from this group to match chosen limit. For example, if ORAT is set, rates of group producers will be recalculated in order to make group oil rate equal to oil limit. Well rates are recalculated with weight coefficients, corresponding to their potentials. Well potential by our definition is well production in absence of rate controls.

In the current version only the limit chosen by control mode will be matched, all other limits are ignored.

Default:

- group name: default group (group of wells with default group in WELSPECS (see 6.10.2)),

- control mode: NONE,

- oil rate limit: no rate or limit,

- water rate limit: no rate or limit,

- gas rate limit: no rate or limit,

- liquid rate limit: no rate or limit,

- group availability: YES,

- groups's share of a higher rate of phase production: no limit.

Example
```
GCONPROD
GROUP11 LRAT 3* 1000 RATE YES /
GROUP2* ORAT 500 3* RATE YES /
/
```

First line sets control for group GROUP11. It will be on liquid rate control, maximum liquid rate for group is 1000. If group rate exceeds this limit, well rates will be recalculated to satisfy this condition. New rates of wells will be proportional to their production potentials (well liquid rate in absence of rate controls); all limits on well BHP and maximum rate etc will be observed.

Second line sets control for all groups with names starting with GROUP2. They will be on oil rate control, maximum oil rate for group is 500. If group rate exceeds this limit, well rates will be recalculated to satisfy this condition. New rates of wells will be proportional to their production potentials (well oil rate in absence of rate controls); all limits on well BHP and maximum rate etc will be observed.

6.10.24   GCONINJE

√	tNavigator	√	eclipse 300	☐	MORE Roxar
√	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword could be specified for a well group or several groups with the same group name root. The following parameters should be specified:

1. group name (or groop name root, i.e. name ending with asterisk), or FIELD (for field control),

2. injected fluid to which group control will apply (WATER or GAS),

3. group control: NONE – no immediate control, RATE – injection rate control, RESV – reservoir volume injection rate (total reservoir volume injection rate of the group meets the parameter 5 of this keyword), REIN – group injection will be equal to group production of the same phase multiplied by reinjection fraction specified in parameter 6, VREP – group injection in reservoir conditions will be equal to group production in reservoir conditions multiplied by voidage replacement fraction specified in parameter 7, or COMP, group injection in reservoir conditions will be equal to group liquid production in reservoir conditions multiplied by voidage replacement fraction specified in parameter 7,

4. surface rate of injected fluid (or limit) (sm^3/day),

5. reservoir rate of injected fluid (or limit) (rm^3/day). If the value is specified by this parameter the phase (parameter 2) is declared the top-up phase. Its target or limitted reservoir volume injection rate will be equal to the value specified here minus the reservoir volume injection rate of other phases. The phase injection rate is calculated to top up the total group injection to the required reservoir volume rate, after allowing for any injection of the other phases. There can be only one top-up phase at any given time in the simulation run,

6. reinjection fraction target (limit), used when group control is set to REIN,

7. voidage replacement target (limit), used when group control is set to VREP or COMP,

8. group availability for higher group production rate target: YES means that group will switch to control from higher group if higher group level limit is violated; NO means that group rate is not influenced by controls from above groups,

9. groups's share of a higher rate of phase injection (phase is set using the second parameter of this keyword). The next parameter of this keyword sets if surface or reservoir volume of the phase will be used,

10. surface or reservoir volume of the phase for the previous parameter of this keyword, current version of tNavigator support only RATE – surface injection rate.

If user specifies group control, and control mode is not NONE, simulator will modify rates of wells (or groups) from this group to match chosen limit. For example, if RATE is set, and phase is WATER, rates of group water injectors will be recalculated in order to make group water injection rate equal to specified limit. Well rates are recalculated with weight coefficients, corresponding to their potentials. Well potential by our definition is well injection in absence of rate controls.

In the current version only the limit chosen by control mode will be matched, all other limits are ignored.

Default:

- group name: default group (group of wells with default group in WELSPECS (see 6.10.2)),

- control mode: NONE,

- rate limit: no rate or limit,

- group availability: YES,

- groups's share of a higher rate of phase injection: no limit.

```
Example
GCONINJE
INJ11 WATER RATE 253.1 2* /
GRP1* WATER VREP 3* 1.1 /
/
```

First line sets control for group INJ11. It is on water injection rate control, the rate is equal to 253.1. If group water injection rate exceeds this limit, water injector rates will be recalculated to satisfy this condition. New rates of wells will be proportional to their injection potentials (well injection rate in absence of rate controls); all limits on well BHP and maximum rate etc will be observed.

Second line sets control for all groups with names starting with GRP1. They will be on voidage replacement control, i.e. injectors will inject in reservoir conditions the same volume, as produced by this group, multiplied by voidage replacement coefficient (1.1). If group injection exceeds this limit, well injection rates will be recalculated to satisfy this condition. New rates of wells will be proportional to their injection potentials (water injection rate in absence of rate controls); all limits on well BHP and maximum rate etc will be observed.

6.10.25   GRUPTREE

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets tree structure for multi-level group control. The tree can consist of an arbitrary number of levels. The field FIELD occupies the top of this tree.

Groups that have other groups as children cannot have wells. (Wells are assigned to groups in the keyword WELSPECS (see 6.10.2)). Thus a group either contains wells (that is a well-group) or has other groups as children (that is a node-group). Groups without a parent group will have a parent group FIELD.

An arbitrary number of data rows can be specified (each row should be ended with a slash /). Each row contains the following data:

1. name of child group;

2. name of parent group (this group is the parent group for the child group in item1).

All data should be terminated with a slash /.

Example
```
GRUPTREE
’GAS1’ ’FIELD’ /
’GAS2’ ’FIELD’ /
’G1’ ’GAS1’ /
’G2’ ’GAS1’ /
’G3’ ’GAS2’ /
’G4’ ’GAS2’ /
/
```

In this example the FIELD (level 0) contains 2 child groups GAS1, GAS2 (level 1). GAS1 contains child group G1 and G2 (level 2), GAS2 - child groups G3 and G4 (level 2).

6.10.26　BRANPROP

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies the extended network branches (extended network model is specified via the keyword NETWORK (see 6.1.42)). This network structure can be different from the structure specified by GRUPTREE (see 6.10.25) (the bottom nodes in the tree should be the same (i.e. well groups)). An extended network should be defined via BRANPROP and NODEPROP (see 6.10.27).

(Section: NETWORK option. Automatic chokes. Compressors. – 2.16.)

An arbitrary number of data rows can be entered (ended with a slash /). One data row consists of the following parameters:

1. name of the branch's downtree node (this node is nearer to the well);

2. name of the branch's uptree node (this node is nearer to the top of tree);

3. VFP table number VFPPROD (see 6.10.19) (for producers) and VFPINJ (see 6.10.18) (for injectors).
   Several pipelines can have the same VFP table if they have similar pressure loss characteristics. If there is no pressure loss in the network branch defined by parameters 1 and 2, a value 9999 should be entered.
   Branches representing chokes (parameter 3 of the keyword NODEPROP (see 6.10.27)), should have a value 9999.
   A value 0 closes the branch (removes it from the network). Node's production (or node's injection) is not added to the network flows.

All data should be terminated with a final slash /.

Example
```
GRUPTREE
'GAS1' 'FIELD' /
'GAS2' 'FIELD' /
'G1' 'GAS1' /
'G2' 'GAS1' /
'G3' 'GAS2' /
'G4' 'GAS2' /
/

BRANPROP
'GASALL' 'FIELD' 2 /
'GAS1' 'GASALL' 2 /
'GAS2' 'GASALL' 2 /
'G1' 'GAS1' 9999 /
'G2' 'GAS1' 9999 /
'G3' 'GAS2' 9999 /
'G4' 'GAS2' 9999 /
/

NODEPROP
'FIELD' 20 /
'GASALL' 1* 'NO' /
'GAS1' 1* 'NO' /
'GAS2' 1* 'NO' /
'G1' 1* 'YES' /
'G2' 1* 'YES' /
'G3' 1* 'YES' /
'G4' 1* 'YES' /
/
```

GRUPTREE (see 6.10.25):
In this example the FIELD (level 0) contains 2 child groups GAS1, GAS2 (level 1). GAS1 contains child group G1 and G2 (level 2), GAS2 - child groups G3 and G4 (level 2).

BRANPROP (see 6.10.26):
Table VFPPROD (see 6.10.19) number 2 corresponds to the branches GASALL – FIELD, GAS1 – GASALL, GAS2 – GASALL.

Branches G1 – GAS1, G2 – GAS1, G3 – GAS2, G4 – GAS2 represent chokes (flag YES is specified by the keyword NODEPROP (see 6.10.27) for nodes G1, G2, G3, G4).

NODEPROP (see 6.10.27):
The terminal node FIELD has terminal pressure 20*barsa*.

6.10.27 NODEPROP

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword defines the extended network node properties (extended network model is specified via the keyword NETWORK (see 6.1.42)). The keyword BRANPROP (see 6.10.26) should previously describe branch properties. An extended network should be defined via BRANPROP (see 6.10.26) and NODEPROP (see 6.10.27).

(Section: NETWORK option. Automatic chokes. Compressors. – 2.16.)

The top node of each network tree (the terminal node) should be a fixed pressure node. Only the fixed pressure node, and other nodes with non-default properties need to be specified here.

An arbitrary number of data rows can be entered (ended with a slash /). One data row consists of the following parameters:

1. node name;

2. fixed pressure for the terminal node *barsa* (if the node is not a terminal node, this parameter should be defaulted, or set negative);

3. flag indicating whether the uptree branch from this node should act as an automatic choke to impose a flow rate limit. (The uptree branch is the one towards the terminal node; the outlet branch for a production network or the inlet branch for an injection network.) An automatic choke controls the flow of a target group by adjusting the pressure drop across the choke. The branch specified via BRANPROP (see 6.10.26) should have a VFP table number 9999, so that its only pressure drop is that of the choke. The terminal node cannot be selected for this purpose, because it has no uptree branch and its pressure is fixed.

   YES – The node's uptree branch acts as an automatic choke. NO – A production target applied to the corresponding group is met by the standard methods of group control.

   Manifold groups, and groups for automatic chokes, should have their rate targets set directly with GCONPROD (see 6.10.23) or indirectly as a share of a higher level group's rate target. In the latter case, that is, if they are subordinate to a higher level group having a production rate target (or a limit that becomes a rate target if violated), they should be given guide rates in keyword GCONPROD (see 6.10.23). Groups subordinate to an automatic choke's target group should not have guide rates. Production wells subordinate to an automatic choke's target group are not subject to guide rate group control.

All data should be terminated with a final slash /.

    Default:

- flag indicating whether the uptree branch from this node should act as an automatic choke to impose a flow rate limit – NO.

Example
```
GRUPTREE
'GAS1' 'FIELD' /
'GAS2' 'FIELD' /
'G1' 'GAS1' /
'G2' 'GAS1' /
'G3' 'GAS2' /
'G4' 'GAS2' /
/

BRANPROP
'GASALL' 'FIELD' 2 /
'GAS1' 'GASALL' 2 /
'GAS2' 'GASALL' 2 /
'G1' 'GAS1' 9999 /
'G2' 'GAS1' 9999 /
'G3' 'GAS2' 9999 /
'G4' 'GAS2' 9999 /
/

NODEPROP
'FIELD' 20 /
'GASALL' 1* 'NO' /
'GAS1' 1* 'NO' /
'GAS2' 1* 'NO' /
'G1' 1* 'YES' /
'G2' 1* 'YES' /
'G3' 1* 'YES' /
'G4' 1* 'YES' /
/
```

GRUPTREE (see 6.10.25):
In this example the FIELD (level 0) contains 2 child groups GAS1, GAS2 (level 1). GAS1 contains child group G1 and G2 (level 2), GAS2 - child groups G3 and G4 (level 2).

BRANPROP (see 6.10.26):
Table VFPPROD (see 6.10.19) number 2 corresponds to the branches GASALL – FIELD, GAS1 – GASALL, GAS2 – GASALL.

Branches G1 – GAS1, G2 – GAS1, G3 – GAS2, G4 – GAS2 represent chokes (flag YES is specified by the keyword NODEPROP (see 6.10.27) for nodes G1, G2, G3, G4).

NODEPROP (see 6.10.27):

The terminal node **FIELD** has terminal pressure $20 barsa$.

6.10.28   NETCOMPA

√ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
√ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword defines automatic compressors or pumps in the extended production network (extended network model is specified via the keyword NETWORK (see 6.1.42)). An extended network should be defined via BRANPROP (see 6.10.26) and NODEPROP (see 6.10.27).

(Section: NETWORK option. Automatic chokes. Compressors. – 2.16.)

Compressor (pump) occupies a branch of the network, and its operation is modelled by changing the VFP table number or ALQ value of the branch.

An arbitrary number of data rows can be entered (ended with a slash /). One data row consists of the following parameters:

1. inlet node name of the branch where the compressor is;

2. outlet node name of the branch where the compressor is;

3. name of group whose production rate this compressor will respond to; The compressor will be activated if this group fails to meet its target production rate set for the phase in parameter 4. If the group does not have a target set, but has a guide rate defined in GCONPROD (see 6.10.23) and is under FLD control from a higher level group, then the compressor is activated when the group cannot make its share of the higher level group's production target.

4. phase whose production rate this compressor responds to.
   OIL, GAS.

5. VFP table number for the branch when the compressor is on. When the compressor is switched on this replaces the table number previously specified in BRANPROP (see 6.10.26), which is applied whenever the compressor is off. 0 – original VFP table number will be unchanged.

6. ALQ – Artificial Lift Quantity to be applied when the compressor is fully on. When the compressor is fully on this replaces the ALQ value previously specified in keyword BRANPROP (see 6.10.26), which is applied whenever the compressor is off. Multi-level compressors operating at an intermediate level has an intermediate value of the ALQ (parameters 9, 10 and 11). The artificial lift quantity may be regarded as the compressor or pump power, according to the definition used when the table was calculated (for example with VFP).

7. rate of gas consumption by the compressor when fully on. This is an Eclipse compatibility field. IGNORED.

8. name of the group from which the compressor's gas consumption should also be extracted. This is an Eclipse compatibility field. IGNORED.

9. compressor type

   - PERM – compressor remains on permanently after being turned on.

   - TEMP – compressor can be turned off manually (for example if the target production rate is reduced). It turns on automatically again as soon as it is needed. In gas field operation model the compressor turns off automatically whenever the contract group's target rate decreases. It turns on automatically again as soon as it is needed.

   - MULT – Similar to TEMP, but with multiple compression levels. Compression is increased one level at a time until the group's target rate is satisfied.

10. number of compression levels in a multi-level compressor.
    This parameter should be specified only if parameter 9 is MULT.
    If there are levels the ALQ and consumption rateat level are:

    $$ALQ_i = ALQ_1 + (ALQ_N - ALQ_1(\frac{i-1}{N-1}))$$

    $$CONS_i = CONS_N * \frac{i}{N}$$

    where ALQ_1 is specified in parameter 11. VFP table number is specified in parameter 5, when the compressor is working at the first level and above.

11. artificial lift quantity at level 1 of multi-level compressor, ALQ_1.
    This parameter should be specified only if parameter 9 is MULT.

12. compressor switching sequence number. This parameter specifies the order in which compressors is turned on if there are two or more compressors corresponding to the same group (parameter 3). If the group cannot satisfy its rate target, the compressor with the lowest sequence number that corresponds to the group is turned on. If the group still cannot satisfy its target when this is fully on, but has other compressors that correspond to it, these other compressors are turned on in increasing order of their sequence number. If two compressors in the network have the same sequence number, they are both turned on at the same time (it is so also if they respond to different groups). In this case it is possible to turn on all compressors simultaneously if the compression is needed anywhere in the field. All multi-level compressors with the same sequence number have their levels increasing simultaneously.

All data should be terminated with a final slash /.
   Default:

   - name of group whose production rate this compressor will respond to – compressor is turned off;

- phase whose production rate this compressor responds to – GAS;

- VFP table number for the branch when the compressor is on – 0.

Example
```
NETCOMPA
'GASALL' 'FIELD' 'FIELD' 'GAS' 2 100 2* 'MULT' 200 1 /
/
```

In this example compressor is in the branch 'GASALL' – 'FIELD', corresponds to the production rate of the group 'FIELD', phase – gas. VFP table number for this branch – 2. Compressor type – MULT. Number of compression levels – 1.

6.10.29   DRSDT

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets maximum rate of increase of solution gas-oil ratio. The data should be terminated with a slash /.

The following parameters should be specified:

1. maximum rate at which the solution gas-oil ratio ($R_{G,O}$) in any grid block is allowed to increase ($sm^3/sm^3/day$);

2. flag: ALL – parameter 1 is applied to all grid blocks, FREE – parameter 1 is applied only to grid blocks that contain free gas.

Default: flag: ALL.

The keyword defines how free gas and undersaturated oil interact in a grid block. If maximum rate – 0, $R_{G,O}$ cannot rise and free gas does not dissolve in undersaturated oil. If maximum rate is large, $R_{G,O}$ rises until the oil is saturated or no free gas remains (this way is if this keyword isn't specified).

Example
```
DRSDT 0.0008/
```

6.10.30   COMPENSATION

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines compensation factor and type for group of wells. The following parameters are to be specified:

1. group name,

2. compensation fraction (between 0 and 1),

3. compensation type.

The only supported in current version compensation type is INTEGRAL. For this type of compensation the total injection rate of all injection wells in the group will be proportional increased or decreased to reach the specified compensation fraction from the total production rate of all production wells in the group. If computed bottom hole pressure for an injection well exceeds limit on BHP, then this well is switched on control by BHP and compensation does not reach the specified compensation fraction.

Default: none

```
Example
COMPENSATION
Group1 1.0 INTEGRAL /
Group2 0.9 INTEGRAL /
/
```

In this example for group Group1 compensation fraction is set to 100% of liquid produced. For group Group2 compensation fraction is set to 90%.

6.10.31   GECON

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines economic limit for group or several groups with the same group name root. The following parameters should be specified:

1. group name (or groop name root, i.e. name ending with asterisk), or FIELD (for field control),

2. lower economic limit of oil rate (if violated, all wells in the group will be shut),

3. IGNORED, this is an Eclipse compatibility field,

4. upper economic limit of water cut,

5. IGNORED, this is an Eclipse compatibility field,

6. IGNORED, this is an Eclipse compatibility field,

7. workover action in case of water cut violation:

   - NONE – do nothing
   - CON – shut in the worst offending perforation,
   - CON+ – shut in worst offending perforation and all below,
   - +CON – the same as CON+,
   - WELL – shut in the worst offending well

Default:

- lower oil rate economic limit — 0,

- upper water cut economic limit — no limit,

- workover action: NONE.

```
Example
GECON
LEFT_SIDE 30 1* 0.95 2* CON /
PROD* 30 4* WELL /
/
```

This example sets limits for several groups. For group LEFT_SIDE minimum possible oil rate is equal to 30, and maximum possible water cut is 95%. If group oil rate falls below 30, all wells in the group will be shut in; if group water cut exceeds 95%, the maximum offending perforation in maximum water cut well will be closed.

For groups with name starting with PROD minimum possible oil rate is also equal to 30, and there is no water cut limit. If calculated oil rate is lower than 30, all wells in the group will be shut in.

6.10.32 WELDRAW

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets maximum drawdown allowed for production wells (single well or group of wells). The data for each well (group) should be terminated with a slash /. At the end of the data one should put an extra slash /. The following parameters are to be specified for each well (group of wells):

1. well name or group name (or groop name root, i.e. name ending with asterisk), or FIELD (to set maximum drawdown allowed for field),

2. maximum drawdown allowed for production well (group of wells),

3. phase; production rate limit for this phase will be calculated at every time step using the maximum drawdown allowed for this well (group) and phase's mobility in corresponding connections: LIQ (water and oil) – if well produces oil, GAS – if well produces gas.

   This formula converts maximum drawdown allowed for the well (group) into maximum production rate for the phase:

   $$Q_{Max} = D_{Max} \sum_i (T_{ci} * M_i)$$

   Q_{Max} – maximum production rate for the phase, D_{Max} – maximum drawdown allowed for the well, T_c – connection transmissibility factor, M – phase's mobility in this connection, i – the sum is over all open connection at current time step.

4. maximum drawdown allowed for the well availbility for well's production potential: YES – means that the well's production potential depends on limits: BHP, THP and drawdown, NO – means that the well's production potential depends on limits: BHP and THP.

Default:

- phase (the third parameter of this keyword) – LIQ,

- the 4th parameter – YES: the well's production depends on limits: BHP, THP and drawdown.

If maximum production rate is calculated from maximum drawdown allowed for the well, the well will be on gas or liquid control (phase is set by 3rd parameter of this keyword). Phase rate limit – Q_{Max}. If the rate limit of this phase is set manually (with the help of keywords WCONPROD (see 6.10.12) or WELTARG (see 6.10.17)), tNavigator will take the minimum of these two values.

> **Example**
> ```
> WELLDRAW
> LEFT_SIDE 280 LIQ NO /
> WELL136 300 GAS YES /
> /
> ```

This example sets maximum drawdown allowed for the group of wells LEFT_SIDE to 280, phase – LIQ (production rate limit for this phase will be calculated at every time step using the maximum drawdown allowed for this group and phase's mobility in corresponding connections), well's production potential doesn't depend on drawdown limit. Maximum drawdown allowed for the well WELL136 is set to 300, phase – GAS, well's production potential depends on drawdown limit.

6.10.33   DATES

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines sequential dates for input and output data. On the next strings date is written in format: DD 'MONTH' YYYY, each date must end with a slash /. Here 'MONTH' can take the following values: JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC, or JLY, which is acceptable alternative to JUL. Dates must be written in ascending order.

Default: none

```
Example
DATES
01 JUL 1984
/
...
DATES
01 JAN 1985
/
...
DATES
01 JUL 1985 /
...
DATES
01 JAN 1986 /
...
DATES
01 JUL 1986 /
...
/
```

This example sets dates for input and output: every half a year between 01 JUL 1984 and 01 JUL 1986.

```
Example
DATES
01 FEB 1985
/
...
DATES
01 MAR 1985
/
WELSPECS
1043 G1 14 10 /
1054 1* 15  8 /
/
COMPDAT
1043 14 10 1 1 OPEN 2* 0.16 3* Z /
1054 15  8 1 1 OPEN 2* 0.16 3* Z /
/
WCONPROD
1043 OPEN LRAT 63.4   0 0 63.4 2* /
1054 OPEN LRAT 59.6 6.4 0 66  2* /
/
DATES
01 MAY 1985
/
WCONPROD
1043 OPEN LRAT 80 0 0 90 2* /
1054 OPEN LRAT 67 9 0 76 2* /
/
DATES
01 JAN 1986
/
DATES
01 JAN 1987
/
```

In this example during development stage two new wells are added (on the 1st of March in 1985) and all appropriate controls are set; in two months (May 1st, 1985) these controls change.

6.10.34   TSTEP

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword explicitly defines sequential time step sizes for input and output data (in days). Time step sizes should be separated by spaces, it is possible to use short form with asterisks and braces. If DATE keyword is used after TSTEP keyword, the user should provide that the date of new time step start is later then the date of the last time step.

Default: none

```
Example
TSTEP
31 30
2*31 2*{30 31}
/
```

This example sets the following simulation time steps: 31 day, 30 days, 31 day, 31 day, 30 days, 31 day, 30 days, 31 day.

```
Example
DATES
01 MAR 1990
/
...
TSTEP
31
/
...
DATES
01 MAY 1990
/
...
TSTEP
31
/
...
```

This example sets the following date sequence: 1st of March, 1990; 1st of April, 1990; 1st of May, 1990, 1st of June, 1990.

6.10.35　WELOPEN

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

　　The keyword shuts or reopens wells and well connections. The data for each well should be terminated with a slash /. The data for all wells should be terminated with a final slash /. To shut or open a well one should specify first and second parameter of the keyword. To change the current status of one connection enter its coordinates using 3-5 parameters. A zero value of parameter 3-5 is considered as any number.

　　The following parameters are to be specified:

1. well name (well number),

2. status (OPEN, STOP, SHUT or AUTO - well connection will open automatically when the value of oil saturation runs up to WELSOMIN (see 6.10.1)),

3. *X* - coordinate of connection,

4. *Y* - coordinate of connection,

5. *Z* - coordinate of connection.

Example
```
WELOPEN
Well1 SHUT /
Well2 OPEN /
Well3 OPEN 0 0 5/
/
```

　　This example shuts Well1, opens Well2, opens all connections of Well3 in layer 5.

6.10.36 RUNCTRL

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify the iterative process parameters. The following options may be used:

1. DTMIN n, minimal allowed time step,

2. DTMAX n, maximal allowed time step,

3. DPLIM n, maximal allowed pressure relative variation,

4. DNLIM n, maximal allowed molar volume relative variation,

5. DVLIM n, maximal allowed pore volume relative variation,

6. TMAXMULT n, allowed maximal time step increasing factor per one time step (must be > 1),

7. TOLNEWT n, maximal admissible residual to finish Newton iterations,

8. TOLVARNEWT n, maximal admissible major variables variance to finish Newton iterations,

9. TOLLIN n, tolerance of linear solver,

10. MINNEWTIT n, minimal number of Newton iterations,

11. MAXNEWTIT n, maximal number of Newton iterations,

12. MAXLINIT n, maximal number of iterations in linear solver,

13. MAXWELLIT n, maximal number of iterations for one well,

14. TOLWELL n, tolerance of linear solver for one well,

15. MAXELAPSED n, maximal time of one calculation step (minutes),

16. MAXWELLDPCHECKS n, maximal number of checks of pressure variation in grid blocks with opened perforations.

Default:

DTMIN	0.1
DTMAX	100.0
DPLIM	0.2
DNLIM	0.2
DVLIM	0.2
TMAXMULT	4.0
TOLNEWT	0.001
TOLVARNEWT	0.001
TOLLIN	0.0001
MINNEWTIT	1
MAXNEWTIT	100
MAXLINIT	1000
MAXWELLIT	8
TOLWELL	0.001
MAXWELLDPCHECKS	0

```
Example
...
DATES
01 JUN 2000
/
RUNCTRL
MAXLINIT 100
MAXNEWTIT 5
TOLLIN 0.00001
TOLNEWT 0.001
/
...
DATES
01 AUG 2000
/
RUNCTRL
DTMAX 10.0
DTMIN 0.0002
/
...
DATES
01 JAN 2001
/
RUNCTRL
DTMAX 1.0
DTMIN 0.00002
TOLLIN 0.00001
/
...
DATES
01 JAN 2003
/
RUNCTRL
MAXELAPSED 2.5
/
...
```

In this example first we set values of 4 parameters – maximum numbers of linear solver and Newton iterations, and target precision for linear solver and for Newton process. On the first of August 2000 new settings are added – for some reason, for example, due to massive well switches, we reduce maximum time step down to 10 (the default is 100) and also reduce minimum time step down to 0.0002. From the first of January 2001 the minimum time step will reduce even more, as well as maximum admissible time step.

From the first of January 2003 the maximal time of one calculation step will be 2,5 minutes.

6.10.37   MULTSIG

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword can be used in dual porosity run 2.20 (DUALPORO (see 6.1.37)) if sigma-factor is set using the keywords SIGMA (see 6.2.53), SIGMAV (see 6.2.54). Sigma-factor is multiplied by the multiplier MULTSIG.

One value for all blocks should be entered. The data should be terminated with a slash /. Different multipliers for grid blocks can be entered using the keyword MULTSIGV (see 6.10.38).

Example
```
MULTSIG 0.3 /
```

6.10.38   MULTSIGV

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword can be used in dual porosity run 2.20 (DUALPORO (see 6.1.37)) if sigma-factor is set using the keywords SIGMA (see 6.2.53), SIGMAV (see 6.2.54). Sigma-factor is multiplied by the multiplier MULTSIGV.

One should enter NX * NY * (NZ/2) values (matrix blocks). The data should be terminated with a slash /. The common multiplier for all grid blocks can be entered using the keyword MULTSIG (see 6.10.37).

Example
```
MULTSIGV 20*0.3 30*0.5/
```

This example sets the multiplier for 50 blocks: for 20 blocks – 0.3, for 30 blocks —- 0.5.

6.10.39   WNNC

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

For simulation of behind-the-casing flow (section 5.4.2) if an option **Behind-the-casing flow** is enable in tNavigator the keywords WNNC (see 6.10.39) and WCONNNC (see 6.10.40) can be used.

- **Manually input** of connection effective multipliers (the keyword WNNC (see 6.10.39)).

- **Automatic calculation** of connection effective multipliers using the data of the keyword WCONNNC (see 6.10.40).

The keyword WNNC can specify for one well a list of pairs of linked blocks and connection effective multiplier for each pair of blocks. Each pair and it's effective multiplier should be followed by a slash /. All data should be terminated with a final slash /.

The following parameters of WNNC should be specified:

1. well name;

2. i_1 – X-coordinate of block with connection;

3. j_1 – Y-coordinate of block with connection;

4. k_1 – Z-coordinate of block with connection;

5. i_2 – X-coordinate of block which is linked with block $[i_1, j_1, k_1]$;

6. j_2 – Y-coordinate of block which is linked with block $[i_1, j_1, k_1]$;

7. k_2 – Z-coordinate of block which is linked with block $[i_1, j_1, k_1]$;

8. γ – connection effective multiplier for blocks $[i_1, j_1, k_1]$ and $[i_2, j_2, k_2]$.

6.10.40　WCONNNC

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

For simulation of behind-the-casing flow (section 5.4.2) if an option **Behind-the-casing flow** is enable in tNavigator the keywords WNNC (see 6.10.39) and WCONNNC (see 6.10.40) can be used.

- **Manually input** of connection effective multipliers (the keyword WNNC (see 6.10.39)).

- **Automatic calculation** of connection effective multipliers using the data of the keyword WCONNNC (see 6.10.40).

The keyword **WCONNNC** can specify for one well a list blocks and K, M for them. Each list of blocks and it's K, M should be followed by a slash /. All data should be terminated with a final slash /.

The following parameters of **WCONNNC** should be specified:

1. well name;

2. i – X-coordinate of block with connection (block also can have the status PATH);

3. j – Y-coordinate of block with connection (block also can have the status PATH);

4. k – coordinate of block with connection (block also can have the status PATH);

5. K – effective permeability along the well bore;

6. M – permeability multiplier along the well bore.

Default: Coordinates i, j, k can be defaulted. In this case all blocks of the well bore will be taken into consideration.

```
Example
WCONNNC
'WU1' 3* 0.01 /
/
```

In this example for all blocks containing the well bore of the well WU1 is specified the value of effective permeability along the well bore – 0.01.

6.10.41   WFRAC

	tNavigator		eclipse 300		MORE Roxar
✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the hydraulic fracture. The data should be terminated with a slash /.

The following parameters should be specified:

1. well name;

2. first connection (or trajectory) coordinate in X direction;

3. first connection (or trajectory) coordinate in Y direction;

4. first connection (or trajectory) coordinate in Z direction;

5. last connection (or trajectory) coordinate in X direction;

6. last connection (or trajectory) coordinate in Y direction;

7. last connection (or trajectory) coordinate in Z direction;

8. azimuth angle (from $0°$ to $360°$);

9. zenithal angle (from $0°$ to $90°$);

10. half of fracture lenght (m);

11. fracture width (m);

12. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

13. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

14. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time).

Default:

- proppant name – disable; infinite permeability along the fracture (pipe);

- flow function name – disable; there is no dependence from flow.

Example
```
WFRAC
'WELL123' 38 426 13 38 426 21 30 0.5 250 5 'propant 16/20'
'func 5' 'LIQ'
/
```

In this example hydraulic fracture if specified the following way: well WELL123, first connection – [38, 426, 13], last connection – [38, 426, 21], azimuth angle – $30°$, zenithal angle – $0.5°$, half of fracture lenght – 250, fracture width – 5, propant 16/20, flow function – func 5, dependence of liquid flow – LIQ.

6.10.42 WFRACL

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the hydraulic fracture for the wells in local grids (section 2.21). The data should be terminated with a slash /.

WFRACL must be used in place of WFRAC (see 6.10.41) to specify the hydraulic fracture for wells in local refined grids, after the wells have been introduced with keywords WELSPECL (see 6.10.3), COMPDATL (see 6.10.7). The keyword data for WFRACL is the same as for WFRAC (see 6.10.41), except for an extra parameter 2 which names the local grid containing the connections specified in the record.

The following parameters should be specified:

1. well name;

2. local grid name (CARFIN (see 6.2.61));

3. first connection (or trajectory) coordinate in X direction;

4. first connection (or trajectory) coordinate in Y direction;

5. first connection (or trajectory) coordinate in Z direction;

6. last connection (or trajectory) coordinate in X direction;

7. last connection (or trajectory) coordinate in Y direction;

8. last connection (or trajectory) coordinate in Z direction;

9. azimuth angle (from $0°$ to $360°$);

10. zenithal angle (from $0°$ to $90°$);

11. half of fracture lenght (m);

12. fracture width (m);

13. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

14. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

15. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time).

Default:

- proppant name – disable; infinite permeability along the fracture (pipe);

- flow function name – disable; there is no dependence from flow.

```
Example
WFRACL
'WELL123' 'LGR2' 38 426 13 38 426 21 30 0.5 250 5 'propant
16/20' 'func 5' 'LIQ'
/
```

In this example hydraulic fracture if specified the following way: well WELL123 (situated in local grid LGR2), first connection – [38, 426, 13], last connection – [38, 426, 21], azimuth angle – 30°, zenithal angle – 0.5°, half of fracture lenght – 250, fracture width – 5, propant 16/20, flow function – func 5, dependence of liquid flow – LIQ.

6.10.43 WFRACP

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the hydraulic fracture. This keyword is the expension of the keyword WFRAC (see 6.10.41). WFRACP is used to specify hydraulic fracture from graphical interface. The data should be terminated with a slash /.

The following parameters should be specified (Hydraulic fracture is in the plane, which contains the well bore, is on the picture below):

1. well name;

2. $i1$ – first connection coordinate in X direction;

3. $j1$ – first connection coordinate in Y direction;

4. $k1$ – first connection coordinate in Z direction;

5. $i2$ – last connection coordinate in X direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $i1 = i2$;

6. $j2$ – last connection coordinate in Y direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $j1 = j2$;

7. $k2$ – last connection coordinate in Z direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $k1 = k2$;

8. azimuth angle (from $0°$ to $360°$);

9. zenithal angle (from $0°$ to $90°$);

10. $l1$ – fraction lenght in one direction from the well bore (m);

11. $l2$ – fraction lenght in second direction from the well bore (m);

12. $h1$ – fraction height in one direction from the well bore (m);

13. $h2$ – fraction height in second direction from the well bore (m);

14. fracture width (m);

15. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

16. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

17. phase (flow function in previous parameter depends on this phase flow) or time dependence (**OIL** – oil, **WAT** – water, **GAS** – gas, **LIQ** – liquid, **TIME** - time);

18. fraction productivity multiplier;

19. X1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified);

20. Y1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified);

21. Z1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified);

22. X2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified);

23. Y2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified);

24. Z2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified).

    Default:

- proppant name – disable; infinite permeability along the fracture (pipe);

- flow function name – disable; there is no dependence from flow;

- fraction productivity multiplier – 1;

- X1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified) – 1;

- Y1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified) – 1;

- Z1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified) – 1;

- X2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified) – NX (see 6.1.12);

- Y2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified) – NY (see 6.1.12);

- Z2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified) – NZ (see 6.1.12).
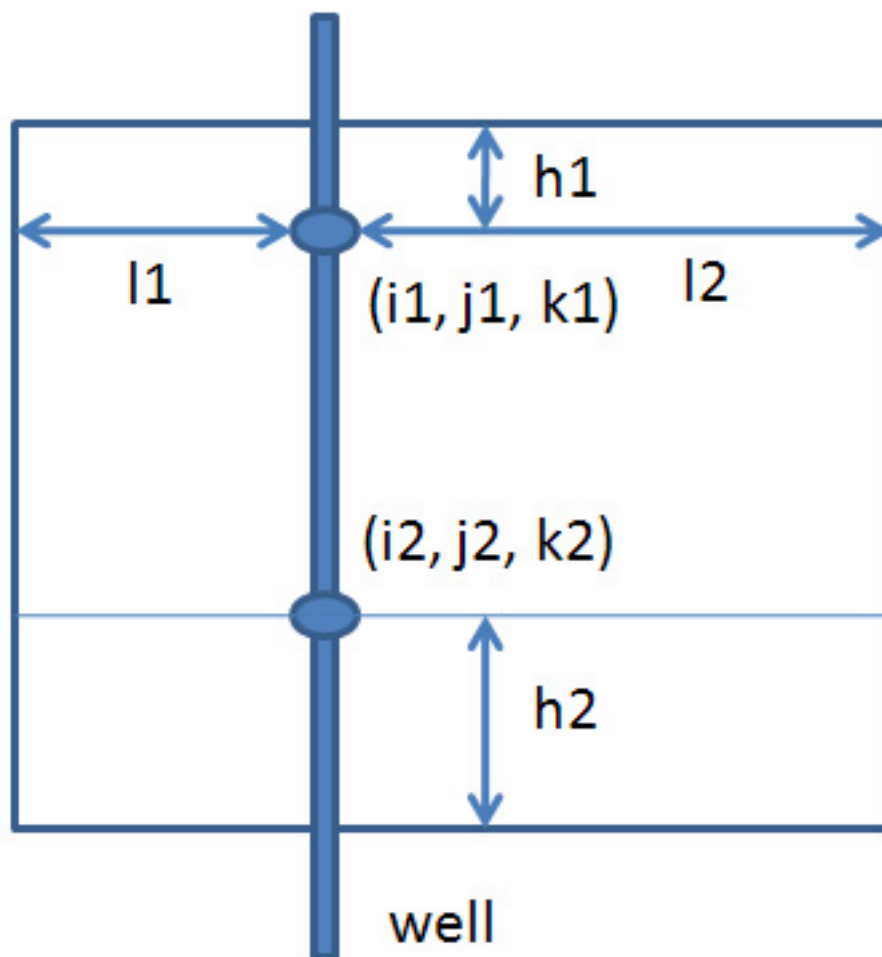
Рис. 2: Hydraulic fracture is in the plane that contains the well bore

Fracture is presented as a part of plane (is visualized in graphical interface), virtual perforations are made in all grid blocks, which this plane crosses.

```
Example
WFRACP
'WELL123' 22 27 1 22 27 8 110 0 1400 1400 0 0 0.02 'propant
12/18' 1* 1* 5 6*
/
```

In this example hydraulic fracture if specified the following way: well WELL123, first connection – [22, 27, 1], last connection – [22, 27, 8], azimuth angle – 110°, zenithal angle – 0°, fracture lenght in one direction – 1400 m, fracture lenght in second direction – 1400 m fracture width – 0.02 m, proppant – propant 12/18, flow function name – disable; there is no

dependence from flow, fraction productivity multiplier - 5, coordinate of bounding box – non specified (i.e. the reservoir).

6.10.44   WFRACPL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the hydraulic fracture for the wells in local grids (section 2.21). This keyword is the expension of the keyword WFRAC (see 6.10.41). WFRACP is used to specify hydraulic fracture from graphical interface. The data should be terminated with a slash /.

WFRACPL must be used in place of WFRACP (see 6.10.43) to specify the hydraulic fracture for wells in local refined grids, after the wells have been introduced with keywords WELSPECL (see 6.10.3), COMPDATL (see 6.10.7). The keyword data for WFRACPL is the same as for WFRACP (see 6.10.43), except for an extra parameter 2 which names the local grid containing the connections specified in the record.

The following parameters should be specified (Hydraulic fracture is in the plane, which contains the well bore, is on the picture below):

1. well name;

2. local grid name (CARFIN (see 6.2.61));

3. $i1$ – first connection coordinate in X direction;

4. $j1$ – first connection coordinate in Y direction;

5. $k1$ – first connection coordinate in Z direction;

6. $i2$ – last connection coordinate in X direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $i1 = i2$;

7. $j2$ – last connection coordinate in Y direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $j1 = j2$;

8. $k2$ – last connection coordinate in Z direction. If hydraulic fracture is in the plane that is perpendicular to the well bore, only one connection should be specified, i.e. $k1 = k2$;

9. azimuth angle (from $0°$ to $360°$);

10. zenithal angle (from $0°$ to $90°$);

11. $l1$ – fraction lenght in one direction from the well bore (m);

12. $l2$ – fraction lenght in second direction from the well bore (m);

13. $h1$ – fraction height in one direction from the well bore (m);

14. *h2* – fraction height in second direction from the well bore (*m*);

15. fracture width (*m*);

16. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

17. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

18. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

19. fraction productivity multiplier;

20. X1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified);

21. Y1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified);

22. Z1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified);

23. X2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified);

24. Y2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified);

25. Z2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified).

Default:

- proppant name – disable; infinite permeability along the fracture (pipe);

- flow function name – disable; there is no dependence from flow;

- fraction productivity multiplier – 1;

- X1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified) – 1;

- Y1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified) – 1;
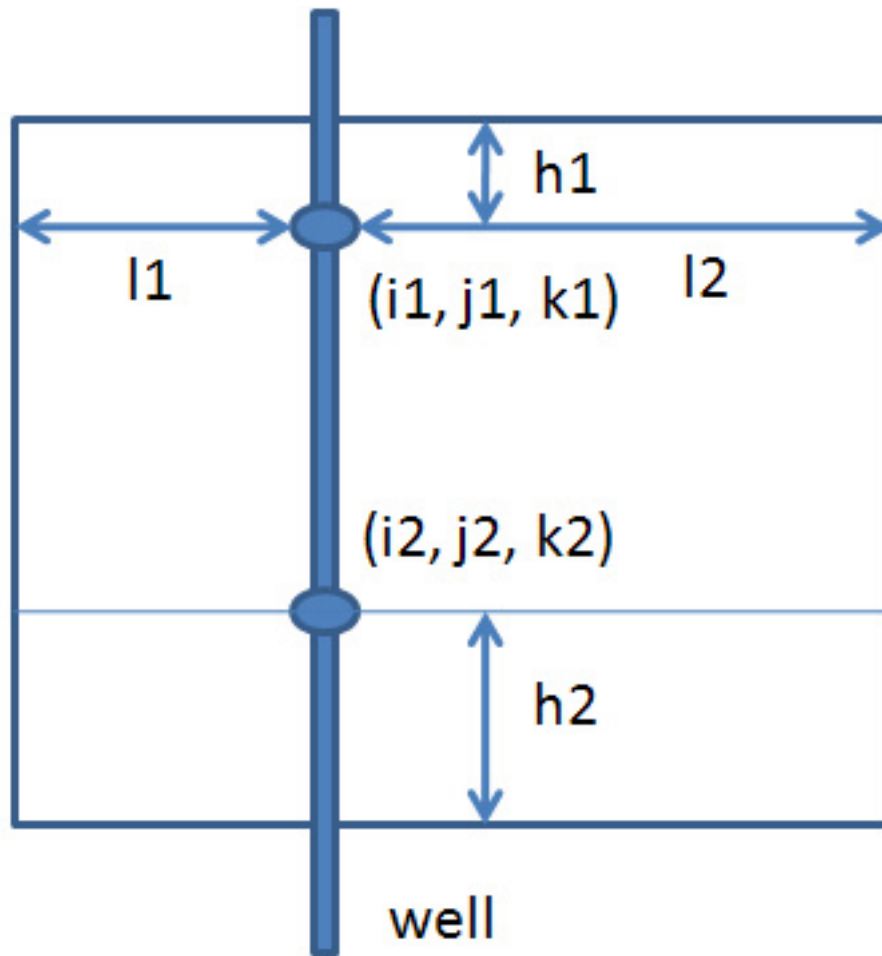
Рис. 3: Hydraulic fracture is in the plane that contains the well bore

- Z1-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified) – 1;

- X2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in X direction should be specified) – NX (see 6.1.12);

- Y2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Y direction should be specified) – NY (see 6.1.12);

- Z2-coordinate of bounding box (the fracture shouldn't be outside this box) (number of layer in Z direction should be specified) – NZ (see 6.1.12).

Fracture is presented as a part of plane (is visualized in graphical interface), virtual perforations are made in all grid blocks, which this plane crosses.

Example
```
WFRACP
'WELL123' 'LGR1' 22 27 1 22 27 8 110 0 1400 1400 0 0 0.02
'propant 12/18' 1* 1* 5 6*
/
```

In this example hydraulic fracture if specified the following way: well **WELL123** (situated in the local grid **LGR1**), first connection – [22, 27, 1], last connection – [22, 27, 8], azimuth angle – 110°, zenithal angle – 0°, fracture lenght in one direction – 1400 m, fracture lenght in second direction – 1400 m fracture width – 0.02 m, proppant – propant 12/18, flow function name – disable; there is no dependence from flow, fraction productivity multiplier - 5, coordinate of bounding box – non specified (i.e. the reservoir).

6.10.45 COMPFRAC

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the hydraulic fracture for connection in the grid layer. The data should be terminated with a slash /.

The following parameters should be specified:

1. well name – well name;

2. i – connection coordinate in X direction. For vertical wells coordinates i, j can be defaulted (the values specified via the keyword COMPDAT (see 6.10.6)). Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC" condition is satisfied";

3. j – connection coordinate in Y direction; For vertical wells coordinates i, j can be defaulted (the values specified via the keyword COMPDAT (see 6.10.6)). Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC" condition is satisfied";

4. k – connection coordinate in Z direction. Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC" condition is satisfied";

5. flag – two values are possible – OPEN, SHUT (open, shut hydraulic fracture). If SHUT is used all the parameters below can be defaulted;

6. azimuth angle (from $0°$ to $360°$);

7. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

8. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

9. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

10. skin – fraction efficiency; specified as skin-factor in classical approximation of hydraulic fracture;
    if skin = 0 (default value) the fraction is not created (this is the same as flag = SHUT)
    if skin < 0 the fraction efficiency and lenght is as big, as the absolute value of skin is;

11. pi – fraction productivity multiplier;

12. mult – fraction lenght multiplier;

13. formula type for fraction lenght – **BLOCK** or **SKIN**. For **BLOCK** type fraction lenght is calculated as the product of skin, mult and block diagonal (root from the sum of squares of DX / DY / DZ (see 6.2.1)). For **SKIN** type fraction lenght is calculated as the product of skin, mult and 50 (there is no dependence of block size).

   Default:

   - proppant name – disable; infinite permeability along the fracture (pipe);

   - flow function name – disable; there is no dependence from flow;

   - skin = 0;

   - pi = 1;

   - mult = 1;

   - formula type for fraction lenght – **BLOCK**.

The keyword creates the hydraulic fracture. The differences with the fracture, created with WFRAC (see 6.10.41), are the following:

   - the fracture is created only in the layer k, fraction height is equal to the block height, in which fraction goes in this layer;

   - fraction lenght and width are calculated as the projection of layer k to the plane OXY, i.e. Z coordinate is not taken into consideration;

   - inflow to the hydraulic fracture belongs to inflow to connection i, j, k.

Example
```
COMPFRAC
'WELL123' 38 426 13 30 'propant 16/20' 'func 5' 'LIQ' -4 3*
/
```

In this example hydraulic fracture is specified the following way: well **WELL123**, connection – [38, 426, 13], azimuth angle – $30°$, propant **16/20**, flow function (dependence of phase **LIQ**) – func **5**, skin = -4.

6.10.46   COMPFRACL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the hydraulic fracture for connection in the grid layer for the wells in local grids (section 2.21). The data should be terminated with a slash /.

COMPFRACL must be used in place of COMPFRAC (see 6.10.45) to specify the hydraulic fracture for connection in the grid layer for wells in local refined grids, after the wells have been introduced with keywords WELSPECL (see 6.10.3), COMPDATL (see 6.10.7). The keyword data for COMPFRACL is the same as for COMPFRAC (see 6.10.45), except for an extra parameter 2 which names the local grid containing the connections specified in the record.

The following parameters should be specified:

1. well name – well name;

2. local grid name (CARFIN (see 6.2.61));

3. i – connection coordinate in X direction. For vertical wells coordinates i, j can be defaulted (the values specified via the keyword COMPDAT (see 6.10.6)). Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

4. j – connection coordinate in Y direction; For vertical wells coordinates i, j can be defaulted (the values specified via the keyword COMPDAT (see 6.10.6)). Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

5. k – connection coordinate in Z direction. Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

6. flag – two values are possible – OPEN, SHUT (open, shut hydraulic fracture). If SHUT is used all the parameters below can be defaulted;

7. azimuth angle (from $0°$ to $360°$);

8. proppant name (the name should be input via the keyword PROPANTNAMES (see 6.5.17));

9. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

10. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

11. skin – fraction efficiency; specified as skin-factor in classical approximation of hydraulic fracture;
    if skin = 0 (default value) the fraction is not created (this is the same as flag = SHUT)
    if skin < 0 the fraction efficiency and lenght is as big, as the absolute value of skin is;

12. pi – fraction productivity multiplier;

13. mult – fraction lenght multiplier;

14. formula type for fraction lenght – BLOCK or SKIN. For BLOCK type fraction lenght is calculated as the product of skin, mult and block diagonal (root from the sum of squares of DX / DY / DZ (see 6.2.1)). For SKIN type fraction lenght is calculated as the product of skin, mult and 50 (there is no dependence of block size).

Default:

- proppant name – disable; infinite permeability along the fracture (pipe);

- flow function name – disable; there is no dependence from flow;

- skin = 0;

- pi = 1;

- mult = 1;

- formula type for fraction lenght – BLOCK.

The keyword creates the hydraulic fracture. The differences with the fracture, created with WFRACL (see 6.10.42), are the following:

- the fracture is created only in the layer k, fraction height is equal to the block height, in which fraction goes in this layer;

- fraction lenght and width are calculated as the projection of layer k to the plane OXY, i.e. Z coordinate is not taken into consideration;

- inflow to the hydraulic fracture belongs to inflow to connection i, j, k.

```
Example
COMPFRACL
’WELL123’ ’LGR1’ 38 426 13 30 ’propant 16/20’ ’func 5’ ’LIQ’
-4 3*
/
```

In this example hydraulic fracture is specified the following way: well WELL123 (situated in local grid LGR1), connection – [38, 426, 13], azimuth angle – $30°$, propant 16/20, flow function (dependence of phase LIQ) – func 5, skin = -4.

6.10.47   MULTPERM

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the multiplication of absolute permeability tensor in grid block (along a vector) by the number. The data should be terminated with a slash /.

The following parameters should be specified:

1. well name – well name;

2. i – block (usually connection) coordinate in X direction. Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

3. j – block (usually connection) coordinate in Y direction. Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

4. k – block (usually connection) coordinate in Z direction. Via ACTIONC (see 6.10.55) all values could be defaulted, i.e. "the value should be taken from the connection for which "ACTIONC"condition is satisfied";

5. flag – two values are possible – YES or NO (multiply/set the initial value of permeability tensor);

6. azimuth angle (from $0°$ to $360°$);

7. zenithal angle (from $0°$ to $90°$);

8. skin – multiplication efficiency; specified as skin-factor in classical approximation of hydraulic fracture;
   if skin = 0 (default value) there is no multiplication (this is the same as flag = NO) if skin < 0 the absolute permeability tensor is multiplied (along the vector with azimuth and zenithal angles) by the number of the absolute value of skin.

Default:

- skin = 0.

```
Example
MULTPERM
'WELL123' 38 426 13 30 45 -4
/
```

In this example: well WELL123, connection – [38, 426, 13], azimuth angle – $30°$, zenithal angle – $45°$, skin = -4.

6.10.48   WPIFUNC

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies for the connection the function of dependence between fraction permeability and phase flow. Efficiency factor (the keyword WPIMULT (see 6.10.10)) is multiplied by this function (5.12). The data should be terminated with a slash /.

The following parameters should be specified:

1. well name;

2. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

3. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

4. connection (or trajectory) coordinate in X direction;

5. connection (or trajectory) coordinate in Y direction;

6. connection (or trajectory) coordinate in Z direction;

7. number of first completion in range (completion numbers are set via COMPLUMP (see 6.10.8)),

8. number of last completion in range (completion numbers are set via COMPLUMP (see 6.10.8)).

Flow function is specified for connection which have: connections with coordinates XYZ, specified by parameters 4-6 of this keyword; number of completion should be between numbers specified by parameters 7-8.

Default:

- flow function name – disable; there is no dependence from flow rate,

- number of first completion in range — any number,

- number of last completion in range — any number.

```
Example
WPIFUNC
'WELL123' 'func 5' 'LIQ' 38 426 13 3 9 /
```

6.10.49 WSKFUNC

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies for the connection the function of dependence between skin-factor and phase flow. Skin-factor (the keyword COMPDAT (see 6.10.6)) is multiplied by this function (5.13). The data should be terminated with a slash /.

The following parameters should be specified:

1. well name;

2. flow function name (the dependence between fracture permeability and phase flow). Phase is specified via the next parameter of this keyword. The function specifies the washing out of the proppant from the fracture. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

3. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

4. connection (or trajectory) coordinate in X direction;

5. connection (or trajectory) coordinate in Y direction;

6. connection (or trajectory) coordinate in Z direction;

7. number of first completion in range (completion numbers are set via COMPLUMP (see 6.10.8)),

8. number of last completion in range (completion numbers are set via COMPLUMP (see 6.10.8)).

Flow function is specified for connection which have: connections with coordinates XYZ, specified by parameters 4-6 of this keyword; number of completion should be between numbers specified by parameters 7-8.

Default:

- flow function name – disable; there is no dependence from flow rate,

- number of first completion in range — any number,

- number of last completion in range — any number.

Example
```
WSKFUNC
'WELL34' 'func 2' 'OIL' 38 426 13 3 9 /
```

6.10.50 WBHZONE

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies the properties of well bottom zone (5.13). The data should be terminated with a slash /.

The following parameters should be specified:

1. well name;

2. radius of well bottom zone $r_{bhz}^{w,l}$ (5.13);

3. flow function name (the dependence between well bottom zone radius and phase flow rate). Phase is specified via the next parameter of this keyword. The function specifies the plugging of the well bottom zone. (keywords FLOWFUNC (see 6.5.19), FLOWFTAB (see 6.5.22), FLOWFNAMES (see 6.5.21));

4. phase (flow function in previous parameter depends on this phase flow) or time dependence (OIL – oil, WAT – water, GAS – gas, LIQ – liquid, TIME - time);

5. permeability of well bottom zone in radial direction $K_{bhz}^{w,l}$ (5.13);

6. permeability of well bottom zone in vertical direction $K^{w,l}$ (5.13);

7. connection (or trajectory) coordinate in X direction;

8. connection (or trajectory) coordinate in Y direction;

9. connection (or trajectory) coordinate in Z direction;

10. number of first completion in range (completion numbers are set via COMPLUMP (see 6.10.8)),

11. number of last completion in range (completion numbers are set via COMPLUMP (see 6.10.8)).

Well bottom zone properties are specified for connections which have: connections with coordinates XYZ, specified by parameters 7-9 of this keyword; number of completion should be between numbers specified by parameters 10-11.

Default:

- radius of well bottom zone $r_{bhz}^{w,l}$ – not specified;

- flow function name – disable; there is no dependence from flow rate,

- phase LIQ – liquid,

- permeability of well bottom zone in radial direction $K_{bhz}^{w,l}$ – not specified;

- permeability of well bottom zone in vertical direction $K^{w,l}$ – not specified;

- number of first completion in range — any number,

- number of last completion in range — any number.

Example
```
WBHZONE
'PROD' 2 'func 2' 'OIL' 10 0.1 4 17 2/
```

6.10.51 ACTION

☑ tNavigator	☑ eclipse 300	☐ MORE Roxar
☑ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword initiates a set of keywords to be processed when a specified field condition is satisfied.

The keyword ACTION (G, R, W) marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a specified condition is satisfied. The keyword ACTION specifies a field condition for triggering an action. Keywords ACTIONG (see 6.10.52), ACTIONR (see 6.10.53), ACTIONW (see 6.10.54) offer greater flexibility, and specify actions triggered by group, region, well conditions respectively. The set of keywords must be terminated with the keyword ENDACTIO. The keywords between ACTION and ENDACTIO are processed when the condition defined in the ACTION keyword is satisfied.

Almost any SCHEDULE section keyword may be placed between ACTION and ENDACTIO, with the exception of timestepping keywords (TSTEP (see 6.10.34), DATES (see 6.10.33), etc.).

Each action initiated with the ACTION keyword is executed only once, at the end of the timestep during which its triggering condition is first satisfied. Each action keyword set must be bracketed by a pair of ACTION (G, R, W) and ENDACTIO keywords, and is distinguished by its action name defined in parameter 1 of the keyword. If an ACTION family keyword is re-entered with the same action name as a previously declared action, the action keyword set is overwritten by the new data.

The following parameters are to be specified (The data should be terminated with a slash /.):

1. action name;

2. quantity to which the triggering condition applies:

  - FOPR – Field oil production rate,

  - FWPR – Field water production rate,

  - FGPR – Field gas production rate,

  - FGOR – Field gas oil ratio,

  - FWCT – Field water cut,

  - FPR – Field average pressure,

  - FOPT – Field oil production cumulative total,

  - FWPT – Field water production cumulative total,

  - FGPT – Field gas production cumulative total,

- FOIR – Field oil injection rate,

- FWIR – Field water injection rate,

- FGIR – Field gas injection rate,

- FOIT – Field oil injection cumulative total,

- FWIT – Field water injection cumulative total,

- FGIT – Field gas injection cumulative total.

3. the operator for the triggering condition:

   - < – Less than;

   - > – Greater than.

4. the value of the triggering condition.

```
Example
ACTION
A1 FGOR > 3.0 /
WELOPEN
WELL1 OPEN /
/
WELTARG
'P*' ORAT 1800 /
/ ENDACTIO
ACTION
A2 FPR < 200 /
END
ENDACTIO
```

In this example there are two action keyword sets. The first action A1 will open the well WELL1, when the field gas oil ratio is greater than 3.0, and set the well oil production rate limits to 1800. The second action A2 will end the run when the average pressure in the field falls below 200.

6.10.52 ACTIONG

✓ **tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword initiates a set of keywords to be processed when a group specifies a specific condition.

The keyword ACTIONG marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated group satisfies a specified condition. The set of keywords must be terminated with the keyword ENDACTIO.

The keywords between the ACTIONG and ENDACTIO keywords are processed at the end of the timestep when the condition defined in the ACTIONG keyword is satisfied by a group named in parameter 2. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied. If a group keyword is placed between ACTIONG and ENDACTIO, instead of entering the name of a specific group in the keyword you may enter a question mark. That applies the keyword data to whichever group (or groups) triggered the action.

Almost any SCHEDULE section keyword may be placed between ACTIONG and ENDACTIO, with the exception of timestepping keywords (TSTEP (see 6.10.34), DATES (see 6.10.33), etc.).

Each action keyword set must be bracketed by a pair of ACTION (G, R, W) and]ENDACTIO keywords, and is distinguished by its action name defined in parameter 1. This name must be unique across all the action keyword family. If an ACTION (G, R, W) keyword is re-entered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The following parameters are to be specified (The data should be terminated with a slash /.):

1. action name;

2. group name (or name root) of the group(s) that triggers this action, or FIELD (for an action triggered by a field quantity). A group name root, enclosed in quotes and ending with an asterisk (*), can be used to make this action's triggering condition apply to more than one group. The action will then be performed if one or more groups having this name root satisfy the triggering condition. The action is performed at most once per timestep, even if more than one group satisfies the triggering condition at the end of the timestep;

3. the quantity to which the triggering condition applies:

   • GOPR — Group oil production rate,

- GPOT — Group oil production cumulative total,
- GOIR — Group oil injection rate,
- GOIT — Group oil injection cumulative total,
- GWPR — Group water production rate,
- GWPT — Group water production cumulative total,
- GWIR — Group water injection rate,
- GWIT — Group water injection cumulative total,
- GGPR — Group gas production rate,
- GGPT — Group gas production cumulative total,
- GGIR — Group gas injection rate,
- GGIT — Group gas injection cumulative total GVPR,
- GVPR — Group voidage production rate,
- GVPT — Group voidage production cumulative total,
- GVIR — Group voidage injection rate,
- GVIT — Group voidage injection cumulative total,
- GLPR — Group liquid production rate,
- GLPT — Group liquid production cumulative total,
- GGOR — Group gas oil ratio,
- GWCT — Group water cut.

4. the operator for the triggering condition.:

- < – Less than;
- > – Greater than.

5. the value of the triggering condition;

6. number of times this action can be triggered. The action is performed once at the end of each timestep while any group named in parameter 2 satisfies the triggering condition, until it has been performed the specified number of times. A number of 10000 or greater is interpreted as infinity;

7. increment to the triggering condition. After each time the action is performed, the specified increment is added to the triggering condition value (initialized in parameter 5). The increment may be negative, if desired.

Default:

- The value of the triggering condition: 1;

- Increment to the triggering condition: 0.

Example
```
ACTIONG
A2 'GROUP2' GOP < 4000.0 /
WELOPEN
WELL5 OPEN /
/
ENDACTIO
```

In this example there is one action keyword set. The action A2 will open the well WELL5, when the oil production rate of group GROUP2 falls below 4000.

6.10.53 ACTIONR

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword initiates a set of keywords to be processed when a region satisfies a specified condition.

The ACTIONR keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated fluid-in-place region satisfies a specified condition. The set of keywords must be terminated with the ENDACTIO keyword.

The keywords between the ACTIONR and ENDACTIO keywords are processed at the end of the timestep when the condition defined in the ACTIONR keyword is satisfied by a region identified in parameters 2 and 3. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

Almost any SCHEDULE section keyword may be placed between ACTIONR and ENDACTIO, with the exception of timestepping keywords (TSTEP (see 6.10.34), DATES (see 6.10.33), etc.).

Each action keyword set must be bracketed by a pair of ACTION (G, R, W) and]ENDACTIO keywords, and is distinguished by its action name defined in parameter 1. This name must be unique across all the action keyword family. If an ACTION (G, R, W) keyword is re-entered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The following parameters are to be specified (The data should be terminated with a slash /.):

1. action name;

2. fluid-in-place region number, whose conditions trigger this action. This should be an integer. Region 0 refers to the whole field. Fluid-in-place regions are defined with the keyword FIPNUM (see 6.4.4);

3. the fluid-in-place region family, to which the specified region belongs. A default (1*) or blank (' ') entry refers to the standard set of fluid-in-place regions defined with the keyword FIPNUM (see 6.4.4). Additional families of fluid-in-place regions may optionally be defined, using the keyword FIP (see 6.4.5).

4. the quantity to which the triggering condition applies:

   - RPR — Region pressure,
   - ROIP — Region oil in place,
   - RWIP — Region water in place,

- RGIP — Region gas in place (in liquid and gas phases).

5. the operator for the triggering condition:

    - < – Less than;
    - > – Greater than.

6. the value of the triggering condition;

7. number of times this action can be triggered. The action is performed once at the end of each timestep while the triggering condition is satisfied, until it has been performed the specified number of times. A number of 10000 or greater is interpreted as 'infinity';

8. increment to the triggering condition. After each time the action is performed, the specified increment is added to the triggering condition value (initialized in parameter 6). The increment may be negative, if desired.

Default:

- The value of the triggering condition: 1;

- Increment to the triggering condition: 0.

```
Example
ACTIONR
ACT7 3 WELRG RPR > 300 10000 /
WELOPEN
WELL3 OPEN /
/
ENDACTIO
```

In this example there is one action keyword set. The action A7 will open the well WELL3, when average pressure in region 3 of the family WELRG exceeds 300.

6.10.54   ACTIONW

✓ tNavigator	✓ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword initiates a set of keywords to be processed when a well satisfies a specified condition.

The ACTIONW keyword marks the start of a set of SCHEDULE section keywords that are to be stored for later processing when a nominated well satisfies a specified condition. The set of keywords must be terminated with the ENDACTIO keyword. The keywords between the ACTIONW and ENDACTIO keywords will be processed at the end of the timestep when the condition defined in the ACTIONW keyword is satisfied by a well named in parameter 2. You can choose whether the action is to be performed only once, or repeatedly each timestep while the triggering condition is satisfied.

Almost any SCHEDULE section keyword may be placed between ACTIONW and ENDACTIO, with the exception of timestepping keywords (TSTEP (see 6.10.34), DATES (see 6.10.33), etc.).

Each action keyword set must be bracketed by a pair of ACTION (G, R, W) and]ENDACTIO keywords, and is distinguished by its action name defined in parameter 1. This name must be unique across all the action keyword family. If an ACTION (G, R, W) keyword is re-entered with the same action name as a previously declared action, the previously declared action is deleted and replaced with the new action.

The following parameters are to be specified (The data should be terminated with a slash /.):

1. action name;

2. well name (or name root) of the well(s) that triggers this action. A well name root, enclosed in quotes and ending with an asterisk (*), can be used to make this action's triggering condition apply to more than one well. The action is then performed if one or more wells having this name root satisfy the triggering condition. The action will be performed at most once per timestep, even if more than one well satisfies the triggering condition at the end of the timestep;

3. the quantity to which the triggering condition applies:

   - WOPR — Well oil production rate,

   - WOPT — Well oil production cumulative total,

   - WOIR — Well oil injection rate,

   - WOIT — Well oil injection cumulative total,

   - WWPR — Well water production rate,

- WWPT — Well water production cumulative total,
- WWIR — Well water injection rate,
- WWIT — Well water injection cumulative total,
- WGPR — Well gas production rate,
- WGPT — Well gas production cumulative total,
- WGIR — Well gas injection rate,
- WGIT — Well gas injection cumulative total,
- WLPR — Well liquid production rate,
- WLPT — Well liquid production cumulative total,
- WVPR — Well voidage production rate,
- WVPT — Well voidage production cumulative total,
- WVIR — Well voidage injection rate,
- WVIT — Well voidage injection cumulative total,
- WGOR — Well gas oil ratio,
- WWCT — Well water cut,
- WBHP — Well bottom hole pressure,
- WTHP — Well tubing head pressure.

4. the operator for the triggering condition:

   - < – Less than;
   - > – Greater than.

5. the value of the triggering condition.;

6. number of times this action can be triggered. The action will be performed once at the end of each timestep while any well named in parameter 2 satisfies the triggering condition, until it has been performed the specified number of times. A number of 10000 or greater is interpreted as infinity;

7. increment to the triggering condition. After each time the action is performed, the specified increment is added to the triggering condition value (initialized in parameter 5). The increment may be negative, if desired.

Default:

- number of times this action can be triggered: 1;

- increment to the triggering condition: 0.

```
Example
ACTIONW
A5 'PRW*' WOPR < 1000.0 /
WELOPEN
PRWELL2 OPEN /
/
ENDACTIO
```

In this example there is one action keyword set. The action A5 will open the well PRWELL2, when the oil production rate of any of the wells whose names begin with PRW, first falls below 1000.

6.10.55 ACTIONC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies the action which takes place if a specified condition is satisfied for the block with connection.

At first parameters should be specified (ended with a slash /). Then operations should be specified – actions which take place if a specified condition is satisfied for the block with connection. At the end of action the keyword ENDACTIO should be entered.

ACTIONC
action name well name $i1$ $i2$ $j1$ $j2$ $k1$ $k2$ quantity operation value times increment /

...
operations
...

ENDACTIO

The following parameters should be specified:

1. action name – name of this action;

2. well name – well name;

3. $i1$ – first connection coordinate in X direction;

4. $i2$ – last connection coordinate in X direction;

5. $j1$ – first connection coordinate in Y direction;

6. $j2$ – last connection coordinate in Y direction;

7. $k1$ – first connection coordinate in Z direction;

8. $k2$ – last connection coordinate in Z direction;

9. quantity – the quantity to which the triggering condition applies:

    - BPR – pressure in the block i, j, k;
    - CSKIN – skin-factor for the connection in block i, j, k.

10. operation – the operator for the triggering condition; the following operators are possible:

    - < – less than value;

- \> – greater than value.

11. value – the value of the triggering condition;

12. times – number of times this action can be triggered;

13. increment – increment to the triggering condition value.

   Default:

   - $i1$ – first connection coordinate in X direction: 1;

   - $i2$ – last connection coordinate in X direction: NX;

   - $j1$ – first connection coordinate in Y direction: 1;

   - $j2$ – last connection coordinate in Y direction: NY;

   - $k1$ – first connection coordinate in Z direction: 1;

   - $k2$ – last connection coordinate in Z direction: NZ;

   - times – number of times this action can be triggered: 1;

   - increment – increment to the triggering condition value: 0.

In actions enumerated in operations one can use the following expressions to identify connection in block (i, j, k) of well well name, for which the condition is satisfied:

   - ? – as parameter well name in operations stands for the well name well name;

   - * – as a number of connectin i, j, k in operations stands for the corresponding value from well well name;

   - * – as a parameter skin in operations stands for skin-factor of the well well name.

Example 1.

```
Example
ACTIONC
'ACT1' '*' 6* BPR > 300 /

COMPFRAC
'?' 3* OPEN 45 3* -4 3*/

ENDACTIO
```

For all connections of all wells: if pressure in block with connection is greater than 300 bar an operation COMPFRAC (see 6.10.45) should be done one time in this block. The fracture with following properties is created: azimuth angle - 45°, infinite permeability along the fracture, there is no dependence from flow, fraction efficiency with skin-factor -4.

Example 2.

```
Example
ACTIONC
'ACT2' '*' 6* BPR < 200 /

COMPFRAC
'?' 3* SHUT 45 3* -4 3*/

ENDACTIO
```

For all connections of all wells: if pressure in block with connection is less than 200 bar an operation COMPFRAC (see 6.10.45) should be done one time in this block. The fracture with following properties is closed: azimuth angle - 45°, fraction efficiency with skin-factor -4.

Example 3.

```
Example
ACTIONC
'ACT3' '*' 6* CSKIN < 0 /

COMPFRAC
'?' 3* OPEN 45 4* 3*/

ENDACTIO
```

For all connections of all wells with nonzero skin-factor: an operation COMPFRAC (see 6.10.45) should be done one time in this block. The fracture with following properties is created: azimuth angle - 45°, infinite permeability along the fracture, there is no dependence from flow, fraction efficiency with skin-factor equal to skin-factor of this connection.

Example 4.

```
Example
ACTIONC
'ACT4' '*' 6* CSKIN < 0 /

MULTPERM
3* OPEN 45 0 1* /
/

ENDACTIO
```

For all connections of all wells with nonzero skin-factor: an operation MULTPERM (see 6.10.47) should be done one time in this block. Absolute permeability tensor (along the vector with azimuth angle $45°$ and zenithal angle $0°$) will be multiplied by skin-factor of this connection.

6.10.56   WLIMTOL

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets tolerance fractures for economic and other limits. The data should be terminated with a slash /.

The tolerance fraction is applied to the following limits:

1. connection, well, group, field economic limits (WECON (see 6.10.20), GECON (see 6.10.31), 6.10.21);

2. group, field maximum rate limits resulting in well workovers or closures (GCONPROD (see 6.10.23)).

Default: Not specified.

If one of the limits above is violated during any time step then at the end of the time step wells switch to the corresponding control. So the limit is violated for one time step before the control was switched. If the limit is violated by more than the tolerance fraction multiplied by the limiting value, the time step is recalculated after the control was switched. Thus, the change of control takes effect from the beginning of the time step during which the limit would otherwise have been violated.

```
Example
WLIMTOL
0.15 /
```

6.10.57 SEPVALS

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword defines the initial separator conditions (first use of the keyword) and changes them during the simulation (next uses of the keyword). The first **SEPVALS** must be followed by the keyword **GSEPCOND** (see 6.10.58), which allocates well groups to separators. If the separator corresponds to a group, all wells of this group use this separator. If the separator's conditions are altered by redefining them via **SEPVALS**, then the oil and gas rates of these wells are transformed to reflect the change in conditions.

The keyword can only be used in "black oil" run.
There must be at least one **TSTEP** (see 6.10.34) or **DATES** (see 6.10.33) between two entries of **SEPVALS**, in order to specify the moment of condition changes.

The keyword can be followed by any number of data rows. Each row should be terminated with a slash /. The data should be terminated with a final slash /. Each row consists of the following parameters:

- separator name; the number of separators should be less or equal to the number of well groups (the 3-rd parameter of the keyword **WELLDIMS** (see 6.1.21));

- formation volume factor of bubble point oil when flashed from reservoir conditions to stock tank conditions through the separator (rm^3/sm^3, rm^3 - reservoir volume, sm^3 - surface volume);

- solution gas-oil ratio of bubble point oil, when flashed from reservoir conditions to stock tank conditions through the separator (rm^3/sm^3, rm^3 - reservoir volume, sm^3 - surface volume).

```
Example
SEPVALS
SEP1A 1.24 0.49 /
SEP1B 1.243 0.50 /
SEP1C 1.252 0.514 /
/
...
GSEPCOND
GROUP1 SEP1A /
GROUP2 SEP1B /
GROUP3* SEP1C /
/
```

In this example there are 3 separators. Well group GROUP1 uses the separator SEP1A, well group GROUP2 uses the separator SEP1B, all wells which name begins with GROUP3 uses the separator SEP1C.

6.10.58 GSEPCOND

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword assigns separators to well groups. Each separator should be specified earlier via SEPVALS (see 6.10.57). If the separator corresponds to a group, all wells of this group use this separator. If the separator's conditions are altered by redefining them via SEPVALS (see 6.10.57), then the oil and gas rates of these wells are transformed to reflect the change in conditions.

The keyword can be followed by any number of data rows. Each row should be terminated with a slash /. The data should be terminated with a final slash /. Each row consists of the following parameters:

- group name (or a first part of name ending with an asterisk), or FIELD (if the separator is at the field level);

- name of the separator associated with this group;

- (and its subordinate groups and wells).

```
Example
GSEPCOND
GROUP1 SEP1A /
GROUP2 SEP1B /
GROUP3* SEP1C /
/
```

In this example well group GROUP1 uses the separator SEP1A, well group GROUP2 uses the separator SEP1B, all wells which name begins with GROUP3 uses the separator SEP1C.

```
Example
GSEPCOND
FIELD SEP1A /
/
```

In this example all wells use only one separator SEP1A.

6.10.59 WDFAC

	tNavigator		eclipse 300		MORE Roxar
✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets well's D-factor (flow-dependent skin for gas).

The keyword can be followed by any number of data rows. Each row should be terminated with a slash /. The data should be terminated with a final slash /.

Each row consists of the following parameters:

1. well name;

2. D-factor for this well (day/sm^3).

Default: D-factor = 0.

Well's D-factor can also be specified via the 12-th parameter of COMPDAT (see 6.10.6). Connection's D-factor is calculated from well's D-factor. Connections D-factor can be entered directly via the 12-th parameter of COMPDAT (see 6.10.6).

```
Example
WDFAC
PR1 1.34E-3 /
PR2 1.42E-3 /
/
```

In this example D-factor is specified for wells PR1, PR2.

6.10.60   WDFACCOR

	tNavigator		eclipse 300		MORE Roxar
✓		✓		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword sets D-factor correlation (flow-dependednt skin factor for gas).

The keyword can be followed by any number of data rows. Each row should be terminated with a slash /. The data should be terminated with a final slash /.

Each row consists of the following parameters:

1. well name;

2. coefficient A in D-factor formula below;

3. power B of permeability of grid block with connection in D-factor formula below;

4. power C of porosity of grid block with connection in D-factor formula below.

Default:

- coefficient A in D-factor formula below – 0;

- power B of permeability of grid block with connection in D-factor formula below – 0;

- power C of porosity of grid block with connection in D-factor formula below – 0.

D-factor correlation formula, the expression for non-Darcy flow (following Dake):

$$D = A * k^B * \phi^C * \frac{k}{h} * \frac{1}{r_w} * \frac{\gamma_G}{\mu_G}$$

,

where

- A, B, C – are specified via this keyword;

- k – effective permeability of grid block with connection (For a vertical well this permeability is calculated as the geometric mean of the X and Y direction permeabilities);

- ϕ – porosity of grid block with connection;

- h – connection length;

- r_w – wellbore radius;

- γ_G – relative density of gas (produced or injected) at surface conditions with respect to air at standard temperature and pressure;

- μ_G – gas viscosity at bottom hole pressure.

For a well connection D-factor is calculated based on the permeability and porosity of the grid block with connection together with the fluid properties of the wellbore.

Well's D-factor can also be specified via the 12-th parameter of COMPDAT (see 6.10.6). Connection's D-factor is calculated from well's D-factor. Connections D-factor can be entered directly via the 12-th parameter of COMPDAT (see 6.10.6).

Example
```
UNITS FIELD /

WDFACCOR
WELLPR1 4.48E-5 -1.018 0 /
/
```

In this example the coefficients for D-factor firmula are set for the well WELLPR1.

6.10.61 WTRACER

☑ **tNavigator**	☑ **eclipse 300**	☐ **MORE Roxar**
☑ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword sets the value of concentration of a tracer in the injection streams of its associated phase. If the tracer isn't present in the list of tracer names in this keyword, concentration is asuumed to be equal to 0.

The keyword is followed by any number of lines. Each line should be terminated with a slash /. The data should be terminated with a final slash /.

Each line consists of the following parameters:

1. well name (each well should be previously declared as injector),

2. tracer name (specified via the keyword TRACER (see 6.5.24); in Eclipse tracer name may consist of up to 3 characters, but in tNavigator tracer name may consist of any number of characters),

3. value of the tracer concentration in the injection stream T_{conc},

4. value of the cumulative tracer factor T_{cum}. Using the cumulative tracer factor the tracer concentration can be specified as a linear function of the total cumulative injection of the well. If the cumulative tracer factor is specified, then the tracer concentration in the injection stream is given by: $T_c = MIN(CI * T_{cum}, T_{conc})$, where CI - total cumulative injection at the previous timestep,

5. group name.

```
Example
WTRACER
302 'B' 1 /
303 'B' 1 /
304 'B' 1 /
305 'B' 1 /
/
```

In this example the concentration of tracer B is equal to 1 for four injectors: 302, 303, 304, 305.

6.10.62 WSALT

√	**tNavigator**	√	**eclipse 300**	☐	**MORE Roxar**
√	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify the concentration of salt in the injection stream of each well. If the keyword WSALT doesn't appear, then concentration values of zero are assumed.

The keyword is followed by any number of data records. Each data record should terminated with a slash /. All data should terminated with a final slash /.

Each data record should consist of the following parameters:

1. well name (each well should be previously declared as injector),

2. the consentration of salt in the injection stream of well (kg/m^3).

Example
WSALT
105 80/
126 80/
/

In this example the consentration of salt in the injection stream of wells 105, 126 is 80 kg/m^3.

6.10.63 WVFPDP

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
✓ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies a pressure adjustment. The adjustment is added to the value of the well BHP obtained by interpolating the VFP tables. (Could be used for matching a well's flow rate at a given THP, by adjusting the effective pressure loss between the bottom hole and the tubing head. A positive pressure adjustment (for production well) increases BHP and desreases a well's production. Negative adjustment improves well's production).

The third parameter of the keyword is a tubing pressure loss scaling factor. Well's BHP obtained from the VFP table adjusts by multiplying the tubing pressure loss (BHP-THP) by this factor. For production well a scaling factor, greater than 1, increases BHP and desreases a well's production.

The keyword is followed by any number of lines. Each line should be terminated with a slash /. The data should be terminated with a final slash /.

Each line consists of the following parameters:

1. well name,

2. pressure adjustment (*bars*),

3. tubing pressure loss scaling factor f_p. Well's BHP will be adjusted to BHP1: $BHP1 = THP + f_p * (BHP_{tab} - THP)$.

Default:

- pressure adjustment – 0,

- tubing pressure loss scaling factor f_p – 1.

Independently of the use of this keyword, well's BHP is automatically adjusted to take account of any difference between its BHP reference depth (5-th parameter of WELSPECS (see 6.10.2)) and reference depth of VFP-table (VFPPROD (see 6.10.19)), by adding or substracting a hydrostatic pressure correction based on the density of the fluid in the wll bore.

```
Example
WVFPDP
302 2.5 /
303 -11.2 /
304 2 1.21 /
/
```

In this example pressure adjustment is specified for wells 302, 303; for the wel 304 both pressure adjustment and tubing pressure loss scaling factor is specified.

6.10.64   PICOND

☑	**tNavigator**	☑	**eclipse 300**	☐	**MORE Roxar**
☑	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword controls the calculations of generalized pseudo-pressure option for modelling the effects of condensate dropout on the fluid mobilities at producing well connections. (This option is activated in gas condensate runs for wells by entering **GPP** as 8-th parameter of the keyword **WELSPECS** (see 6.10.2)). The data should be terminated with a slash /.

The following parameters should be specified:

1. the maximum interval between pressure quadrature points below the dew point pressure in the calculation of the generalized pseudo-pressure intefral (*bars*),

2. the maximum interval between pressure quadrature points above the dew point pressure in the calculation of the generalized pseudo-pressure intefral (*bars*),

3. damping coefficient *PPDAMP* (from 0 to 1) for the blocking factor. It provides a means of damping oscillations that may result from the explicit calculations of the blocking factor β (which is calculated at the beginning of each time step), by averaging it with its value from the previous time step according to the formula: $\beta = PPDAMP * \beta_{calc} + (1 - PPDAMP) * \beta_{previous}$, where β_{calc} - calculated value at the time step, $\beta_{previous}$ - value from previous time step.

Defauls:

- the maximum interval between pressure quadrature points below the dew point pressure: $4 * p_a tm$ ($p_a tm$ - atmospheric pressure),

- the maximum interval between pressure quadrature points above the dew point pressure: for **e300** data format – $10 * p_a tm$ ($p_a tm$ - atmospheric pressure), for **e100** data format – 0,

- damping coefficient *PPDAMP* – 1.

Example
```
PICOND
28 0.1 0.7/
```

In this example there are: the maximum interval between pressure quadrature points below the dew point pressure - 28 *bars*, the maximum interval between pressure quadrature points above the dew point pressure - 0.1 *bars*, damping coefficient *PPDAMP* - 0.7.

6.10.65   WPAVE

✓	**tNavigator**	✓ **eclipse 300**	☐ **MORE Roxar**
✓	**eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

This keyword controls the calculation of well block average pressures. These averages represent the average pressure of the grid blocks containing connections to a given well, and optionally their adjacent and diagonal neighbors also, weighted according to either the connection transmissibility factors or the grid block pore volumes. Well block average pressures can also can be written to the .res file via the keywords **WBP**, **WBP4**, **WBP5** and **WBP9**. The averages are used for reporting purposes only, and will not affect any other results.

If this keyword is not present, all items assume their default values, giving a connection factor weighted average, evenly weighted between the inner blocks and the outer ring of neighbors. The depth correction uses the wellbore density and only grid blocks associated with currently open well connections are included in the average. The data should be terminated with a slash /.

The following parameters should be specified:

1. the weighting factor $F1$ between the inner block and the outer ring of neighboring blocks, in the connection factor weighted average. If the value lies between 0.0 and 1.0, the average is calculated by the formula:

   When $F1 \geq 0$, the average block pressure for each connection k is the weighted average of the inner block pressure $P_{i,k}$ (i.e. the block containing the connection) and the average of the pressures $P_{o,k}$ in the 4 or 8 blocks surrounding it:

   $$\bar{P}_k = F1 P_{i,k} + (1 - F1) \frac{\sum_{o,k} P_{o,k}}{N_{o,k}}$$

   The value 1.0 gives total weighting to the inner blocks, containing the well connections. The value 0.0 gives total weighting to the 4 or 8 blocks neighboring each inner block.

   A value of $F1 < 0$ is used to indicate that the pressure of the inner block and its outer ring of neighboring blocks should be averaged according to their pore volumes. The average block pressure for each connection is calculated by the formula:

   When $F1 < 0$, the average block pressure for each connection k is the average of the pressures in the inner block $P_{i,k}$ and in each of the 4 or 8 blocks surrounding it $P_{o,k}$, weighted according to their pore volumes $V_{i,k}$ and $V_{o,k}$.

   $$\bar{P}_k = \frac{V_{i,k} P_{i,k} + \sum_{o,k} V_{o,k} P_{o,k}}{V_{i,k} + \sum_{o,k} V_{o,k}}$$

The pressure in each individual grid block $P_{i,k}$ or $P_{o,k}$ is corrected to the well's bottom hole reference depth according to the option selected in parameter 3. The number of surrounding blocks $N_{o,k}$ is 4 for 4-block and 5-block averages (**WBP4**, **WBP5**) and 8 for 9-block averages (**WBP9**). The configuration is shown in figure below. 4- and 5-block averages use the four immediate neighbors (n) of the connecting grid block. 9-block averages use in addition the 4 diagonal neighbors (d). The inner block is ignored in 4-block averages (**WBP4**). The number of neighbors is smaller if the well is situated on the edge of the grid or adjacent to an inactive cell. The neighbors are selected in the plane perpendicular to the direction of penetration of the connection (see parameter 13 of keyword **COMPDAT** (see 6.10.6)). Thus for horizontal wells, the neighbors are in a vertical plane.

For 1-block averages (WBP) $F1$ is effectively 1.0, whatever the value entered here.

2. the weighting factor $F2$ between the connection factor weighted average and the pore volume weighted average, which is used in the formula below. The value should lie between 0.0 and 1.0. The value 1.0 gives a purely connection factor weighted average, and 0.0 gives a purely pore volume weighted average.

   The well block average pressure P_w for a given well is a weighted combination of the connection factor weighted average pressure $P_{w,cf}$ and the pore volume weighted average pressure $P_{w,pw}$:

$$\bar{P}_w = F2\bar{P_{w,cf}} + (1 - F2)\bar{P_{w,pv}}$$

   Connection factor weighted average – this is the average over connections of the average block pressure P_k at each connection k, weighted according to the connection transmissibility factors T_k:

$$\bar{P_{w,cf}} = \frac{\sum_k T_k \bar{P}_k}{\sum_k T_k}$$

   Pore volume weighted average – this is simply the average depth-corrected pressure P_j in the selected set of grid blocks j, weighted by their pore volumes V_j:

$$\bar{P_{w,pv}} = \frac{\sum_j V_j P_j}{\sum_j V_j}$$

3. depth correction flag. This flag controls how the grid block pressures are corrected to the well's bottom hole reference depth (parameter 5 of the keyword **WELSPECS** (see 6.10.2)).

   - **WELL** – The hydrostatic head is calculated using the density of the fluid in the wellbore at the well connections,
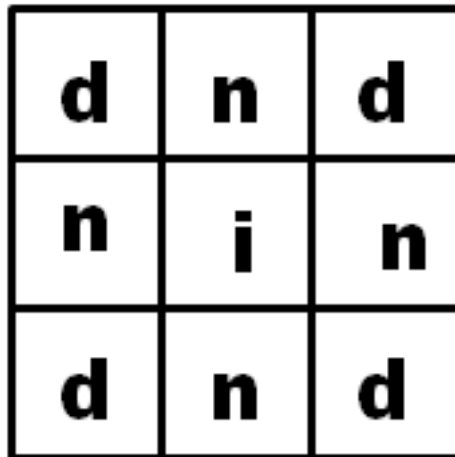
Рис. 4: Grid block configuration in well block average pressure calculations

- **RES** – The hydrostatic head is calculated using a representative density for the fluid in the reservoir. This density is calculated by averaging over fluid density for all the grid blocks associated with the well's connections. The average over phases is weighted by the phase saturation, and the average over grid blocks is weighted by pore volume. Whether the averaging is performed over all grid blocks with declared connections to the well or only those with currently open connections is determined by parameter 4 of this keyword.

- **NONE** – Grid block pressures are not depth corrected.

The wellbore fluid density is set to zero whenever the well is shut. Thus if **WELL** is selected there is a discontinuity in the reported pressure average when the well's status changes between shut and open/stopped.

4. well connection flag. This flag controls whether the grid blocks associated with all the well's declared connections contribute to the average pressure, or just those associated with the currently open connections.

- **OPEN** Only grid blocks associated with currently open connections are included in the averaging calculation.

- **ALL** Grid blocks associated with all currently defined connections (whether open or closed) are included in the averaging calculation.

If **OPEN** is selected there is a discontinuity in the reported pressure average whenever new connections are opened or existing ones are closed. This may be avoided by selecting **ALL** and defining all the well's connections at the start of the run (whether initially open or closed).

Default:

- the weighting factor $F1$ – 0.5,

- the weighting factor $F2$ – 1,

- depth correction flag – WELL,

- well connection flag – OPEN.

Example
```
WPAVE
0.2 1.0 /
```

In this example is given a connection factor weighted average, with increased weighting to the outer blocks.

Example
```
WPAVE
1* 0.0 1* ALL /
```

In this example is given a purely pore volume weighted average over all the blocks containing open or closed well connections and their neighbors.

6.10.66 WRFT

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
✓ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword sets output of well RFT data. The following data will be written to RFT file: pressure, saturation and depth for each grid block in which a well has a connection. An arbitrary number of parameters can be entered. Each parameter should be followed by a slash /. All data should be terminated with a final slash /.

The following parameters should be entered:

1. well name (or number) or well mask, which defines a subset of wells (for example, $*$ – all wells, *PROD$*$* – specifies all wells with names starting from *PROD*).

If the keyword doesn't have any parameters, well RFT data will be output whenever a well is first opened.

```
Example
WRFT
'WELL3*'/
'PROD11'/
/
```

6.11 Inflow from aquifer

tNavigator 3.2 supports the following type of aquifers:

- numerical aquifer. Is set via the keywords AQUCON (see 6.11.9), AQUNUM (see 6.11.8);

- constant-flux aquifer (analytic aquifer). Is set via the keywords AQUFLUX (see 6.11.2), AQUANCON (see 6.11.7);

- Fetkovich aquifer (analytic aquifer). Is set via the keywords AQUFETP (see 6.11.4), AQUANCON (see 6.11.7), AQUFET (see 6.11.3);

- Carter-Tracy aquifer (analytic aquifer). Is set via the keywords AQUTAB (see 6.11.6), AQUANCON (see 6.11.7), AQUCT (see 6.11.5).

Brine option is supported for aquifers (BRINE (see 6.1.33)) (salt concenttation is set via keywords AQUFETP (see 6.11.4), AQUFET (see 6.11.3), AQUCT (see 6.11.5)).

6.11.1 AQUDIMS

	tNavigator		eclipse 300		MORE Roxar
√	tNavigator	√	eclipse 300	☐	MORE Roxar
√	eclipse 100	☐	CMG imex	☐	CMG stars

This keyword sets dimensions for aquifers. The data should be terminated with a slash /. The following parameters should be entered:

1. maximum number of lines of numerical aquifer data entered via keyword AQUNUM (see 6.11.8); this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically;

2. maximum number of lines of connection data for numerical aquifers entered via keyword AQUCON (see 6.11.9); this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically;

3. maximum number of influence tables for Carter-Tracy aquifers, entered via the keyword AQUTAB (see 6.11.6), (if there are any user supplied tables AQUTAB (see 6.11.6) this number should be 2 or greater, because first table is default); this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically;

4. maximum number of rows in a Carter-Tracy aquifer influence table; this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically;

5. maximum number of analytic aquifers in the model; this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically;

6. maximum number of grid blocks connected to any single analytic aquifer; this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically.

7. maximum number of aquifer lists; this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically.

8. maximum number of analytic aquifers in any single aquifer list; this is an Eclipse compatibility field. (IGNORED) In tNavigator number of lines does not have to be specified because the tables are allocated dynamically.

Example
```
AQUDIMS
0 0 6 100 100 100 0 0 /
```

6.11.2 AQUFLUX

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Defines constant flux from aquifer. This keyword may be used in schedule section. Having defined aquifer, connections of grid blocks and aquifer should be defined by means of AQUANCON (see 6.11.7).

Below the following data should be input for each aquifer (terminated with a slash /):

1. aquifer number (positive number),

2. aquifer flux, inflow rate per unit area ($sm^3/day/m^2$),

3. initial salt concentration in the aquifer (kg/m^3) (this parameter can be entered only if Brine option is active BRINE (see 6.1.33)).

All data should be terminated with a final slash /.

The water flow rate into a grid block from a constant flux aquifer is calculated using the formula:

$$Q_w = F_w * A * M$$

where F_w – the flux (parameter 2 of this keyword), A – the area of the connected block face, M – aquifer influx multiplier (parameter 10 of the keyword AQUANCON (see 6.11.7)).

```
Example
UNITS
METRIC
...
AQUFLUX
1 10 /
2 20 /
/
```

In this example two aquifers are set, inflow per square meter from first aquifer will be equal to 10 cubic meters a day, and from second aquifer — 20 square meters. To calculate resulting inflow to grid block, simulator will multiply this number on aquifer influx coefficient multiplier, defined in AQUANCON (see 6.11.7), and then by block face area.

6.11.3 AQUFET

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify properties of Fetkovich aquifer and it's connections with grid blocks. The keyword is used to connected only one reservoir face with one aquifer (if an aquifer is connected to several faces – use the keywords AQUFETP (see 6.11.4), AQUANCON (see 6.11.7)).

Below the following data should be input (terminated with a slash /):

1. datum depth (m),

2. initial pressure on datum depth ($bars$),

3. initial water volume in aquifer (sm^3),

4. total aquifer compressibility (cumulative compressibility of water and rock) ($1/bars$),

5. aquifer productivity index ($sm^3/day/bars$),

6. number of PVTW (see 6.5.4) table, describing properties of water in aquifer,

7. i-coordinate of start of cube connected with aquifer,

8. i-coordinate of end of cube connected with aquifer,

9. j-coordinate of start of cube connected with aquifer,

10. j-coordinate of end of cube connected with aquifer,

11. k-coordinate of start of cube connected with aquifer,

12. k-coordinate of end of cube connected with aquifer,

13. index denoting which face will be connected with aquifer; possible values — $I-$ (face $I=1$), $I+$ (face $I=NX$), $J-$ (face $J=1$), $J+$ (face $J=NY$), $K-$ (face $K=1$), $K+$ (face $I=NZ$) (NX, NY, NZ are set via DIMENS (see 6.1.12)),

14. initial salt concentration in the aquifer (kg/m^3) (this parameter can be entered only if Brine option is active BRINE (see 6.1.33)).

Default:

- initial salt concentration in the aquifer – $0\,kg/m^3$.

The aquifer is connected with grid blocks that are within the specified cube and have no active neighboring block on the side facing the aquifer.

The aquifer productivity index is apportioned between the connecting grid blocks in proportion to the area of their connecting faces.

Example
```
AQUFET
5800 2500 1.7E7 1.0E-6 20.0 1 1 7 1 7 3 3 'K-'
/
```

6.11.4 AQUFETP

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Defines inflow from aquifer calculated according to Fetkovich model. Having defined aquifer, connections of grid blocks and aquifer should be defined by means of AQUANCON (see 6.11.7).

Below the following data should be input:

1. aquifer number (positive number),

2. datum depth (m),

3. initial pressure on datum depth ($bars$),

4. initial water volume in aquifer (sm^3),

5. total aquifer compressibility (cumulative compressibility of water and rock) ($1/bars$),

6. aquifer productivity index ($sm^3/day/bars$),

7. number of PVTW (see 6.5.4) table, describing properties of water in aquifer,

8. initial salt concentration in the aquifer (kg/m^3) (this parameter can be entered only if Brine option is active BRINE (see 6.1.33)).

Each data line should be terminated with a slash /. All data should be terminated with a final slash /.

   Default:

- initial salt concentration in the aquifer – $0\,kg/m^3$.

```
Example
UNITS
METRIC
...
AQUFETP
1 2000 250 100000 1e-5 20 1/
2 2000 250 100000 1e-5 40 1 /
/
```

In this example two aquifers are defined, flow per unit area from second aquifer will be twice higher because it has twice higher productivity index, for the same volume (100000 m^3) and same initial pressure on depth 2000 m, equal to 250 bars.

6.11.5 AQUCT

	tNavigator		eclipse 300		MORE Roxar
✓		☐		☐	
✓	eclipse 100	☐	CMG imex	☐	CMG stars

Defines inflow from aquifer calculated according to Carter Tracy model. Having defined aquifer, connections of grid blocks and aquifer should be defined by means of AQUANCON (see 6.11.7).

Below the following data should be input:

1. aquifer number (positive number),

2. datum depth (m),

3. initial pressure on datum depth ($bars$),

4. aquifer permeability (mD),

5. aquifer porosity,

6. total aquifer compressibility (cumulative compressibility of water and rock) ($1/bars$),

7. reservoir outer radius, i.e. aquifer inner radius (m),

8. aquifer thickness (m),

9. angle of influence (angle subtended by mutual boundary of reservoir and aquifer), degrees,

10. number of PVTW (see 6.5.4) table, describing properties of water in aquifer,

11. AQUTAB (see 6.11.6) table number, cotrolling dependence of pressure on dimensionless time,

12. initial salt concentration in the aquifer (kg/m^3) (this parameter can be entered only if Brine option is active BRINE (see 6.1.33)).

Each data line should be terminated with a slash /. All data should be terminated with a final slash /.

Default:

- aquifer porosity: 1.0;

- angle of influence: 360 degrees;

- PVTW (see 6.5.4) table number: 1;

- AQUTAB (see 6.11.6) table number: 1;

- initial salt concentration in the aquifer – $0\,kg/m^3$.

Example
```
UNITS
METRIC
...
AQUCT
1 2000 250 100 0.3 1e-5 1000 100 90 1 2* /
2 2000 250 100 0.3 1e-5 1000 100 45 1 2* /
/
```

In this example two aquifers are defined, with different angle (90 and 45 degrees), same porosity and permeability (0.3 and 100), compressibility ($1e-5$), thickness (100), radius (1000) and same initial pressure on depth 2000 m, equal to 250 bars.

6.11.6 AQUTAB

	tNavigator		eclipse 300		MORE Roxar
√		√			
√	eclipse 100		CMG imex		CMG stars

The keyword defines an influence function tables for Carter-Tracy aquifers. (the number of tables in set via AQUDIMS (see 6.11.1)). Each table should be terminated with a slash /.

The first table cannot be modified (default table). First table is specified and reserved for the constant terminal rate case for an infinite aquifer as given by Van Everdingen and Hurst. Tables entered via this keyword are numbered from 2.

One table row should contain the following parameters:

1. dimensionless time (positive and increasing down the column),

2. corresponding dimensionless pressure (positive).

```
Example
AQUTAB
0.012 0.167
0.045 0.223
0.1 0.3
0.17 0.38
0.2 0.45
0.27 0.48
0.3 0.5
0.4 0.58
0.5 0.62
0.6 0.68
0.7 0.73
/
```

6.11.7   AQUANCON

	tNavigator		eclipse 300		MORE Roxar
✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

Used to define connections of analytic aquifers (AQUFLUX (see 6.11.2), AQUFETP (see 6.11.4), AQUCT (see 6.11.5), AQUFET (see 6.11.3)) and grid blocks. User inputs table, each line defines cube of grid blocks. Each block will be connected with analytic aquifer by the face indicated for whole cube. The data should be terminated with a slash /.

The following data is input:

1. aquifer number,

2. i-coordinate of start of cube connected with aquifer,

3. i-coordinate of end of cube connected with aquifer,

4. j-coordinate of start of cube connected with aquifer,

5. j-coordinate of end of cube connected with aquifer,

6. k-coordinate of start of cube connected with aquifer,

7. k-coordinate of end of cube connected with aquifer,

8. index denoting which face will be connected with aquifer; possible values — $I-$ (face in direction opposite to X axis direction), $I+$ (face in direction parallel to X axis), $J-$, $J+$ (the same but for axis Y), $K-$, $K+$ (the same but for axis Z),

9. aquifer influx coefficient, determines what fraction of total outflow from aquifer will flow into current block; for constant flux aquifer defined by AQUFLUX (see 6.11.2) keyword, this coefficient is not used. By default it is equal to area of face connected with aquifer. If influx coefficient is specified several times for the same cell, the values are summed. If a zero influx coefficient is specified, this makes cell influx coefficient equal to zero,

10. influx coefficient multiplier; for constant flux aquifer, specified by AQUFLUX (see 6.11.2) keyword, when multiplier is specified, grid block face area changes, being multiplied on it,

11. whether connections from faces, connected with active blocks, are allowed; possible values – YES, NO. By default grid block will be connected with aquifer only if it's face is connected with inactive block or if block has no neighbours from this face.

Default:

- aquifer influx coefficient: grid block area,

- influx coefficient multiplier: 1,

- allow connections from faces connected with inactive blocks: NO

Example
```
DIMENS
10 17 3
/
...
AQUANCON
1 1 10 1 1 1 3 J- 3* /
1 1 1 1 17 1 3 I- 3* /
/
```

In this example connections with aquifer number 1 are specified. Two side faces of reservoir will have water influx, and cells on reservoir arris will have double inflow, because they all have two faces connected, in $I-$ direction and in $J-$ direction.

6.11.8 AQUNUM

✓	**tNavigator**	✓	**eclipse 300**	☐	**MORE Roxar**
✓	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword assigns a numerical aquifer to a block (aquifer is set via the keyword AQUCON (see 6.11.9)). Each line is a grid block representing a numerical aquifer. Each line should be terminated with a slash /. All data should be terminated with a final slash /.

One line should consist of the following parameters:

1. aquifer number,

2. i-coordinate of grid block representing aquifer;

3. j-coordinate of grid block representing aquifer;

4. k-coordinate of grid block representing aquifer;

5. cross-section area (m^2) (may be larger than real cross-section area of the block ijk);

6. length (m) (may be larger than real length of the block ijk);

7. porosity;

8. permeability (mD);

9. aquifer depth (m) (may be larger than real depth of the block ijk);

10. initial pressure ($barsa$);

11. PVT table number for aquifer;

12. saturation table number for aquifer.

Default:

- porosity – value for block ijk;

- aquifer depth – value for block ijk;

- PVT table number for aquifer – PVTNUM (see 6.4.1) value for block ijk;

- saturation table number for aquifer – PVTNUM (see 6.4.1) value for block ijk.

Example
```
AQUNUM
1 1 1 5 120000 18500 1* 350.0 4* /
1 1 1 6 120000 18500 1* 350.0 4* /
/
```

In this example a two-block aquifer is specified, with default porosity, depth, initial pressure, PVT table number and saturation table number.

6.11.9   AQUCON

	tNavigator		eclipse 300		MORE Roxar
✓		✓			
✓	eclipse 100		CMG imex		CMG stars

Used to define connections of numerical aquifers (AQUNUM (see 6.11.8)) and grid blocks. User inputs table, each line defines cube of grid blocks. Each block will be connected with numerical aquifer by the face indicated for whole cube. The data should be terminated with a slash /.

The following data is input:

1. aquifer number,

2. i-coordinate of start of cube connected with aquifer,

3. i-coordinate of end of cube connected with aquifer,

4. j-coordinate of start of cube connected with aquifer,

5. j-coordinate of end of cube connected with aquifer,

6. k-coordinate of start of cube connected with aquifer,

7. k-coordinate of end of cube connected with aquifer,

8. index denoting which face will be connected with aquifer; possible values — $I-$ (face in direction opposite to X axis direction), $I+$ (face in direction parallel to X axis), $J-$, $J+$ (the same but for axis Y), $K-$, $K+$ (the same but for axis Z),

9. transmissibility multiplier (the calculated transmissibility will be multiplied by a user defined multiplier),

10. transmissibility option (0 or 1) The transmissibility between the numerical aquifer and grid block is calculated via the formula:

$$T = \frac{CDARCY}{(\frac{1}{T_a}) + (\frac{1}{T_c})}$$

T_c – component of transmissibility from the grid block;
T_a – component of transmissibility from the aquifer (the formula depends on this parameter of the keyword).

$$T_c = \frac{K_c * A_c * NTG}{D_c}$$

K_c – block permeability in the appropriate direction,
A_c – block face area,
NTG – block net-to-gross ratio if the cell face X or Y,

D_c – distance from the block center the face.

If 0 option:

$$T_a = \frac{K_a * A_a}{D_a}$$

K_a – aquifer permeability (entered via AQUNUM (see 6.11.8)),

A_a – aquifer area (entered via AQUNUM (see 6.11.8)),

D_a – half of aquifer length (entered via AQUNUM (see 6.11.8)).

If 1 option:

$$T_a = \frac{K_a * A_c}{D_a}$$

11. whether connections from faces, connected with active blocks, are allowed; possible values – YES, NO. By default grid block will be connected with aquifer only if it's face is connected with inactive block or if block has no neighbours from this face.

Default:

- transmissibility multiplier – 1,

- transmissibility option – 0,

- allow connections from faces connected with inactive blocks: NO.

Example
```
AQUCON
1 4 4 1 1 2 6 I+ 1.0 /
/
```

In this example connections with aquifer number 1 are specified. The following blocks are connected: 4-th column of X-Z section in the layers 2-6. The face I+ – connections will be made if the corresponding blocks in column 5 are inactive or model size in X-direction is 4 blocks (is specified via first parameter of DIMENS (see 6.1.12)).

7   Keywords compatible with tNavigator and CMG IMEX

For best adaptation of user experience the keyword notations are chosen to be close as much as possible to the most common simulators:

- Eclipse (c) Schlumberger,

- IMEX (c) Computer Modelling Group Ltd,

- STARS (c) Computer Modelling Group Ltd,

- MORE (c) Roxar.

Eclipse compatible keywords are red. For example: TABDIMS (see 6.1.13). Index of Eclipse compatible keywords – 11.

CMG compatible keywords are pink. For example: TEMR (see 8.4.4). Index of CMG compatible keywords – 12.

MORE compatible keywords are green. For example: IDATe (see 9.1.5). Index of MORE compatible keywords – 14.

The keywords which RFD uses only in tNavigator are blue. For example: REACCONC (see 6.8.46). Index of RFD compatible keywords – 13.

In the description of each keyword there is a table where a simulator is ticked off if this keyword is compatible with this simulator.

tNavigator reads keyword notations of these simulators and converts them into its inner data notations.
This section describes all keywords which can be used in tNavigator:

- tNavigator keywords;

- CMG IMEX keywords;

This description pointed out if there are parameters of the keyword which are ignored by tNavigator or which use is different from CMG.

For convenience keyword are grouped in several sections similar to CMG imex sections.

- Data entry system (7.1)

- Input/Output Control (7.2)

- Reservoir description (7.3)

- Component properties (7.4)

- Rock-Fluid data (7.5)

- Initial conditions (7.6)

- Numerical methods control (7.7)

- Well and recurrent data (7.8)

All keyword names are case insensitive. Values of parameters usually follow on the next string. If keyword value is not specified by user the simulator uses a default value if it is defined or an error message is issued.

7.1   Data entry system

7.1.1   MATRIX

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

The keyword is used after keywords which set grid properties. The keyword specifies that properties correspond to matrix blocks.

In dual porosity run (DUALPORO (see 6.1.37)) both MATRIX and FRACTURE (see 7.1.2) are used. FRACTURE (see 7.1.2) specifies fracture properties.

Example
```
PERMI MATRIX ALL
25.5188 25.841 26.0421 26.0878 25.9532 25.6303 25.1359
24.5015 23.7728
```

This example sets X-direction permeabilities for 9 matrix blocks.

7.1.2   FRACTURE

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	✓	CMG imex	☐	CMG stars

The keyword is used after keywords which set grid properties. The keyword specifies that properties correspond to fracture blocks.

In dual porosity run (DUALPORO (see 6.1.37)) both FRACTURE and MATRIX (see 7.1.1) are used. MATRIX (see 7.1.1) specifies matrix properties.

Example
```
POR FRACTURE ALL
0.114087 0.114805 0.115251 0.115352 0.115054 0.114336
0.113228 0.11179
```

This example sets porosity for 9 fracture blocks.

7.1.3 CON

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword is used after the keyword, if constant value array is entered. After CON one should specify the value (which is equal to all array elements).

```
Example
DIFRAC CON 0.1
PB MATRIX CON 6500
```

The keyword sets the distances between fractures (matrix block sizes) in X-direction – DIFRAC – are equal to 0.1. Bubble point pressure for matrix blocks – PB – is equal to 6500.

7.2   Input/Output Control

7.2.1 TITLE1

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	√	**CMG imex**	☐	**CMG stars**

The keyword is used to specify model name (this name can consist of letters and numbers). It has an Eclipse compatible analogue TITLE (see 6.1.1).

Example
TITLE1 'Model number 1'

This example sets the model name Model number 1.

7.2.2   INUNIT

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword is used to specify units system. The data can be read by tNavigator in following units:

- SI,

- METRIC,

- FIELD.

It has an Eclipse compatible analogue UNITS (see 6.1.10).

Example
```
INUNIT FIELD
```

The keyword sets FIELD units system.

7.3 Reservoir Description

7.3.1   ZCORN

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword enables the depths of each corner of each grid block (8 corners) to be separetely specified.

It has an Eclipse compatible analogue ZCORN (see 6.2.33). Data input is identical with this keyword.

7.3.2 COORD

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword is used to specify coordinate lines in Z-direction.

It has an Eclipse compatible analogue COORD (see 6.2.32). Data input is identical with this keyword.

7.3.3   DUALPOR

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

The keyword sets dual porosity run. One should specify matrix block properties and fracture block properties using keywords MATRIX (see 7.1.1) and FRACTURE (see 7.1.2). It has an Eclipse compatible analogue DUALPORO (see 6.1.37).

Example
```
DUALPOR
```

7.3.4 SHAPE

√ tNavigator	☐ eclipse 300	☐ MORE Roxar	
☐ eclipse 100	√ CMG imex	☐ CMG stars	

The keyword sets the type of shape factor which will be used in calculation of matrix-fracture flows in dual porosity run.

There are two possible types:

- GK (Gilman and Kazemi) Matrix-fracture flow in one block is proportional to transmissibility:

$$\frac{20}{3} * k * (\frac{1}{lx} + \frac{1}{ly} + \frac{1}{lz})^2 * MV$$

  k – permeability, lx, ly, lz – distances between fractures in X, Y and Z directions, entered using keywords DIFRAC / DJFRAC / DKFRAC (see 7.3.5), MV – matrix volume.

- WR (Warren and Root) Matrix-fracture flow in one block is proportional to transmissibility:

$$4 * k * (\frac{1}{lx^2} + \frac{1}{ly^2} + \frac{1}{lz^2}) * MV$$

Current version of Navigator support only GK type.

It has an Eclipse compatible analogue VISCD (see 6.1.40). VISCD sets that the viscous displacement option will be used (matrix-fracture flows), but doesn't specify the type of shape-factor. Eclipse compatible analogue – sigma-factor (keywords SIGMA (see 6.2.53), LTOSIGMA (see 6.2.55)).

Example
```
SHAPE GK
```

7.3.5   DIFRAC / DJFRAC / DKFRAC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

The keyword sets the distances between fractures (matrix blocks sizes) in X, Y and Z directions. The keywords can be used if DUALPOR (see 7.3.3) is enable. Matrix-fracture flows are calculated using this keyword (one can observe formula in the description of the keyword SHAPE (see 7.3.4)).

It has an Eclipse compatible analogue LX / LY / LZ (see 6.2.50).

7.3.6 NULL

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

The keyword sets if block is active or inactive (doesn't participate in the run).

- 0 – inactive block,

- 1 – active block.

Default: all blocks are active.

It has an Eclipse compatible analogue ACTNUM (see 6.2.22).

Example
```
NULL MATRIX 22*1 3*0
```

This example sets first 22 matrix blocks active, next 3 - inactive. However inactive block is set only for matrix (MATRIX (see 7.1.1)), flows can appear for fracture part FRACTURE (see 7.1.2).

7.3.7 POR

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	✓ CMG stars

The keywors sets porosity values (between 0 and 1). Reference pressure at which these porosities are given, is specified using the keyword PRPOR (see 8.3.4).

It has an Eclipse compatible analogue PORO (see 6.2.17).

```
Example
POR MATRIX ALL
0.114087 0.114805 0.115251 0.115352 0.115054 0.114336
0.113228 0.11179
POR FRACTURE ALL
0.110115 0.108315 0.106498 0.104755 0.103144 0.10169 0.100393
0.0992395
```

This example sets porosity for 8 blocks (values are different for matrix and fracture blocks).

7.3.8   CPOR / PRPOR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

Two keywords specify a rock compressibility coefficient (CPOR) and a reference pressure (PRPOR). Also porosity is specified at this reference pressure (POR (see 7.3.7)).

These keywords have an Eclipse compatible analogue ROCK (see 6.5.12). Reference pressure – first parameter of ROCK, rock compressibility coefficient – second parameter of ROCK.

```
Example
PRPOR FRACTURE 17820
PRPOR MATRIX 17820
CPOR FRACTURE 1e-6
CPOR MATRIX 1e-7
```

This example sets equal reference pressures for matrix and fracture parts and different rock compressibility coefficients.

7.3.9   PERMI / PERMJ / PERMK

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☑ **CMG imex**	☐ **CMG stars**

These keywords specify absolute permeabilities in X (PERMI), Y (PERMJ), Z (PERMK) directions. In dual porosity run (DUALPOR (see 7.3.3)) one should enter permeabilities for matrix blocks (MATRIX (see 7.1.1)) and fracture blocks (FRACTURE (see 7.1.2)). MATRIX permeabilities are used in calculations of matrix-fracture flows and matrix-matrix flows (in dual permeability run). FRACTURE permeabilities are used in calculations of fracture-fracture flows.

Three keywords have an Eclipse compatible analogue PERMX / PERMY / PERMZ (see 6.2.7).

```
Example
PERMI MATRIX ALL
25.5188 25.841 26.0421 26.0878 25.9532 25.6303 25.1359
24.5015 23.7728
PERMI FRACTURE ALL
2551.88 2584.1 2604.21 2608.78 2595.32 2563.03 2513.59
2450.15 2377.28
```

This example sets absolute permeabilities for 9 matrix and fracture blocks.

7.3.10 NETGROSS

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets net to gross values of grid blocks. The same number of values as number of blocks must be specified.

It has an Eclipse compatible analogue NTG (see 6.2.18).

Example
```
NETGROSS FRACTURE CON 0.32
NETGROSS MATRIX CON 0.32
```

This example sets net fo gross values of matrix and fracture grid blocks equal to 0.32 (dual porosity model).

7.3.11 PINCHOUTARRAY

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets pinched out blocks and not pinched out blocks (blocks with small void volume are not used in the calculations). One should specify the same number of values as the number of blocks in the model.

- 0 – block is pinched out,

- 1 – block is not pinched out.

This keyword has an Eclipse compatible analogues PINCH (see 6.2.42), PINCHREG (see 6.2.45), PINCHNUM (see 6.2.46), PINCHXY (see 6.2.43).

Example
```
PINCHOUTARRAY CON 1
```

This example sets that no one block is pinched out.

7.4   Component properties

7.4.1 MODEL

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword specifies a type of model. tNavivigator read the following types (the keyword doesn't have in influence on the simulation):

- BLACKOIL - oil, gas, water;

- OILWATER - two-phase model, without gas phase;

- MISCD - pseudo-miscible model (solution gas always remain in solution);

- MISNCG - pseudo-miscible model (injected gas has the same composition as solution gas);

- POLY - polymer model (oil, gas, water and polymer);

- POLYOW - polymer model, withoit gas phase;

- API-INT;

- API-INTO;

- GASWATER - gas-water model, without gas phase.

Three keywords have an Eclipse compatible analogue OIL (see 6.1.27), WATER (see 6.1.29), GAS (see 6.1.28), VAPOIL (see 6.1.30), DISGAS (see 6.1.31).

Example
```
MODEL BLACKOIL
```

This example specifies the black oil model.

7.4.2 PVT

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets PVT properties of oil and gas. The data should be entered the following way:

PVT BG (EG of ZG) number

BG – the gas formation volume factor will be used (EG – gas expansion factor, ZG – gas compressibility factor). BG is egual to the volume of gas at reservoir conditions divided by volume of gas at surface conditions, EG is egual to the volume of gas at surface conditions divided by volume of gas at reservoir conditions.

number - PVT region number (the following table is specified for this PVT region).

The table consists of arbitrary numder of lines (two or more). Pressure (the first parameter) should increase down the column. Each line has 8 parameters:

1. the bubble point pressure,

2. gas-oil ratio of saturated oil with buble point pressure specified by the 1-st parameter,

3. the formation volume factor of saturated oil at the bubble point pressure,

4. Bg – the gas formation volume factor will be used (Eg – gas expansion factor, Zg – gas compressibility factor, if EG or ZG is specified after PVT),

5. the viscosity of saturated oil at the bubble point pressure,

6. the viscosity of gas at the bubble point pressure,

7. oil compressibility,

8. gas-oil interfacial tension.

The keyword has an Eclipse compatible analogue PVCO (see 6.5.5), PVDG (see 6.5.6).

Example
```
PVT BG 1
101.325 0.418766947 1.00121067 1.242456434 594.9490888
0.0124995 4.35E-06
527.904 2.185316651 1.006289406 0.235088062 587.1696078
0.0125406 4.35E-06
```

This example specifies PVT table for 1 PVT region (gas-oil interfacial tension isn't specified - default value will be used).

7.4.3 DENSITY

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets oil, water and gas density. To enter phase density one should add phase name OIL, GAS or WATER after DENSITY.

The keyword has an Eclipse compatible analogue DENSITY (see 6.5.15).

Example
```
DENSITY OIL 948.2
DENSITY WATER 1001.48
```

This example sets oil density equal to 948.2 and water density - 1001.48.

7.4.4 BWI / CW / REFPW / CVW / VWI

	tNavigator		eclipse 300		MORE Roxar
√		☐		☐	
☐	eclipse 100	√	CMG imex	☐	CMG stars

These keywords specify for one PVT region the following data:

- BWI - water formation volume factor,

- CW - water compressibility,

- PEFPM - reference pressure,

- CVW - water viscosibility,

- VWI - water viscosity.

If there are more then one PVT region, these five items should be specified for every PVT region.

Five keywords have an Eclipse compatible analogue PVTW (see 6.5.4). 2-nd parameter of PVTW corresponds to BWI, 3-rd – CW, 1-st – REFPW, 5-th – CVW, 4-th – VWI.

```
Example
BWI 1.0111
CVW 0
CW 8.64711e-006
REFPW 20000
VWI 0.613465
```

This example sets water formation volume factor equal to 1.0111, water compressibility - 8.64711e-006, reference pressure - 20000, water viscosibility - 0, water viscosity - 0.613465.

7.4.5 PTYPE

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

The keyword should be followed by one integer for every grid block specifying the PVT region to which it belongs. This keyword should be entered after all PVT tables for all PVT regions.

The keyword has an Eclipse compatible analogue PVTNUM (see 6.4.1).

Example
```
PTYPE FRACTURE CON 1
PTYPE MATRIX CON 1
```

This example sets 1 PVT region (CON specifies an array, all elements of this array are equal to 1).

7.5 Rock-Fluid data

7.5.1 ROCKFLUID

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword starts the "Rock-Fluid data" section (7.5).

Example
```
ROCKFLUID
```

7.5.2   RPT

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	√ CMG imex	√ CMG stars

The keyword should be entered after the keyword ROCKFLUID.
RPT sets the saturation function region number. After this number water-oil (SWT (see 7.5.3))
and liquid-gas (SLT (see 7.5.4)) permeability tables are entered. Then RTYPE (see 7.5.5)
specifies for each grid block the number of saturation function region number to which it
belongs.

The data should be specified the following way:
RPT number COPY oldnumber STONE1 (STONE2),

where

- number – saturation function region number;

- COPY oldnumber – additional option: initializes saturation function region number
  which has the same rock-fluid properties as the saturation function region oldnumber
  (that properties were specified above for oldnumber region);

- STONE1 (STONE2) – additional option: allows to use Stone model in relative
  permeability calculations: STONE1 or STONE2.

In the description of the keyword SLT (see 7.5.4) there is a common example for the
keywords RPT, SWT (see 7.5.3) and SLT (see 7.5.4).

7.5.3   SWT

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	✓ CMG stars

The keyword specifies relative permeability table for water-oil system for one saturation function region (RPT (see 7.5.2) sets saturation function region number).

The table consists of arbitrary numder of lines (two or more). Water saturation (the first parameter) should increase down the column. Each line has 4 parameters:

1. SW (water saturation) (this is argument value for functions below)

2. KRW (water permeability) (this is function k_{rwo} in 2.6)

3. KROW (oil permeability) (this is function k_{row} in 2.6)

4. PCOW (oil-water capillary pressure) (this is function P_{cow} in 2.12.2)

The keyword has an Eclipse compatible analogue SWOF (see 6.6.1).

In the description of the keyword SLT (see 7.5.4) there is a common example for the keywords RPT, SWT (see 7.5.3) and SLT (see 7.5.4).

7.5.4 SLT

	tNavigator		eclipse 300		MORE Roxar
√					
	eclipse 100	√	CMG imex	√	CMG stars

The keyword specifies relative permeability table for gas-oil system for one saturation function region (RPT (see 7.5.2) sets saturation function region number).

The table consists of arbitrary numder of lines (two or more). Liquid saturation (the first parameter) should increase down the column. Each line has 4 parameters:

1. SL (liquid saturation) (this is argument value for functions below)

2. KRG (gas permeability) (this is function k_{rgo} in 2.6)

3. KROG (oil permeability) (this is function k_{rog} in 2.6)

4. PCOG (oil-gas capillary pressure) (this is function P_{cog} in 2.12.1)

The keyword has an Eclipse compatible analogue SGOF (see 6.6.2).

```
Example
RPT 1
SWT
0.202 0 1
0.317 0.003 0.413
0.413 0.011 0.119
0.721 0.063 0.011
0.753 0.121 0
SLT
0.585 0.122 0
0.721 0.089 0.003
0.854 0.031 0.042
0.923 0.01 0.171
0.99 0 1
RPT 2
SWT
0.0 0.0 1.00 0
1 1.0 0.00 0
SLT
0.01 1.0 0.0 0.0
0.99 0.0 1.0 0.0
```

This example sets relative permeability tables SWT (see 7.5.3), SLT (see 7.5.4) for two saturation function regions (RPT 1, RPT 2).

7.5.5   RTYPE

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	√ CMG imex	√ CMG stars

The keyword should be followed by one integer for every grid block specifying the saturation function region to which it belongs. This keyword should be entered after the keyword RPT (see 7.5.2) and tables SWT (see 7.5.3), SLT (see 7.5.4) for all saturation function regions.

The keyword has an Eclipse compatible analogue SATNUM (see 6.4.2).
The keyword also has stars compatible analogue KRTYPE (see 8.5.1).

Example
```
RTYPE MATRIX ALL 1 1 2 2 2 2 3 3 3 3 3 3 3 3 3 2 2 2
RTYPE FRACTURE CON 4
```

In this example matrix blocks (MATRIX) belong to saturation function regions number 1, 2 and 3; fracture blocks (FRACTURE) belong to saturation function region number 4.

7.6   Initial conditions

7.6.1 INITIAL

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword starts the section "Initial conditions" (6.9).

Example
```
INITIAL
```

7.6.2   VERTICAL

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

This option sets that pressures are obtained from hydrostatic equation and saturations – from capillary pressure tables.

The option has these suboptions:
`VERTICAL DEPTH_AVE WATER_OIL EQUIL`

`DEPTH_AVE` - block saturation is determined as average saturation over the depth interval streched over the grid block.
`WATER_OIL` - perform graviry-capillary equilibrium initialization of a reservoir initially containing no gas.
`EQUIL` - during the simulation a pressure correction is added to each phase thus the reservoir initially is in gravitational equilibrium. Saturations are taken as average saturations over depth. Hence gravitational equilibrium isn't ectablished just by setting saturations from capillary pressure tables.

7.6.3 PB

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☑ **CMG imex**	☐ **CMG stars**

The keyword sets initial buble point pressure for each grid block.

The keyword has an Eclipse compatible analogue PBUB (see 6.9.18).

Example
```
PB MATRIX CON 1200
PB FRACTURE CON 1200
```

This example sets initial buble point pressure for matrix (MATRIX) and fracture (FRACTURE) blocks equal to 1200. (CON (see 7.1.3) specifies an array, all elements of this array are equal.)

7.6.4   DATUMDEPTH

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets the datum depth for calculations of depth corrected pressures.
One should spesify the same number of keywords DATUMDEPTH and values as the number of FIP regions.

One can use this keywords with suboption INITIAL: the corrected datum pressures will be calculated using the initial equilibrium pressure distribution.

The keyword has an Eclipse compatible analogue DATUM (see 6.9.22) (for whole reservoir) and DATUMR (see 6.9.23) (for several FIP regions).

Example
```
DATUMDEPTH 2500 INITIAL
```

The example sets datum depth 2500.

Example
```
DATUMDEPTH 2500 INITIAL
DATUMDEPTH 1500
DATUMDEPTH 3125
```

This example sets datum depths for 3 FIP regins.

7.7 Numerical methods control

7.7.1 NUMERICAL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

The keyword starts the section "Numerical methods control" (7.7).

Example
```
NUMERICAL
```

7.8 Well and recurrent data

7.8.1   RUN

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword starts the section "Well and recurrent data" (7.8).

Example
RUN

7.8.2   DATE

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword sets the date and time when the well change occurs. This date should be entered in the following format:
YYYY MM DD
YYYY - year (integer), MM - month (integer), DD - day (real number). If the well change occures at noon the day may be entered as DD.5.

This keyword DATE should be entered immediately after RUN (see 7.8.1) to denote the date of simulation start (an Eclipse compatible analogue is START (see 6.1.7)).
If DATE is used two times and there are well changes between them, these well changes are enable since the first DATE.

The keyword has an Eclipse compatible analogue DATES (see 6.10.33).

Example
```
DATE 1973 4 17.5
```

This example sets the date: April 17, 1973, at noon.

7.8.3 WELL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

This keyword introduces a new well, defining information on its name and coordinates. The data should be specified in the following format:

WELL well-number (well-name) (VERT ibl jbl) (ATTACHTO group-name)

well-number (well-name) – well number (or name); one or two these parameters can be entered.

One can use the suboptions:

- VERT ibl jbl – the well is vertical; ibl – bottom hole or well head coordinates in X direction (IW) and jbl – bottom hole or well head coordinates in Y direction (JW),

- ATTACHTO group-name – name of the group to which this well belongs.

The keyword has an Eclipse compatible analogue WELSPECS (see 6.10.2) (four parameters of WELSPECS are the same).

Example
```
WELL 2 'ProducerL'
WELL 7 VERT 17 23
```

Here two wells are defined: well number 2 'ProducerL' and vertical well number 7, it's bottom hole is situated at $X = 17$ and $Y = 23$.

7.8.4 PRODUCER

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	√	**CMG imex**	☐	**CMG stars**

The keyword specifies a producer (first a well should be entered using the keyword WELL (see 7.8.3)).

The data should be specified in the following format:

PRODUCER well-number (well-name)

One should enter the well number and (or) well name.
The keyword is analogous to the first parameter of an Eclipse compatible keyword WCONPROD (see 6.10.12).

Example
```
WELL 7
PRODUCER 7
```

The example spesifies the well number 7 as producer.

7.8.5 SHUTIN

	tNavigator		eclipse 300		MORE Roxar
√					
	eclipse 100	√	CMG imex		CMG stars

The keyword indicates that a well is shut in (first a well should be entered using the keyword WELL (see 7.8.3), PRODUCER (see 7.8.4)).

The data should be specified in the following format:

SHUTIN well-number (well-name)

One should enter the well number and (or) well name.

The keyword has an Eclipse compatible analogue WELOPEN (see 6.10.35) (allows to shut in a well or connection).

Example
```
WELL 7
PRODUCER 7
DATE 1973 4 17
SHUTIN 7
```

This example shuts in a well number 7 on April 17, 1973.

7.8.6 OPERATE

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

This keyword sets the well controls and action to be done if one of controls is violated. (First a well should be entered using the keyword WELL (see 7.8.3), PRODUCER (see 7.8.4).) The data should be entered in the following format:

OPERATE MAX (or MIN) well-control value action

where:

MAX (or MIN) - maximum (MAX) or minimum (MIN) control is specified using value;
well-control - well control:

- STL – liquid rate control (oil and water) (LRAT in Eclipse compatible keywords),

- STO – oil rate control (ORAT in Eclipse compatible keywords),

- STG – gas rate control (GRAT in Eclipse compatible keywords),

- STW – water rate control (WRAT in Eclipse compatible keywords),

- BHP – bottom hole pressure control; minimum bottom hole pressure for producers must be specified, otherwise it will be taken 101.3 kPa (BHP in Eclipse compatible keywords),

- WHP – tubing head pressure control (THP in Eclipse compatible keywords),

- BHF – reservoir liquid rate control (oil, water, gas) (RESV in Eclipse compatible keywords).

value - control value;
action - action to be done if this control is violated: CONT – well will switch to the violated control.

The keyword has an Eclipse compatible analogue WCONPROD (see 6.10.12) (for producers), WCONINJE (see 6.10.14) (for injectors).

Example
```
WELL 7
PRODUCER 7
OPERATE MAX STL 42 CONT
OPERATE MIN BHP 250 CONT
```

This example sets for producer number 7 maximum liquid rate control – 42, minimum bottom hole pressure control – 250. If one of these controls is violated, well will switch to the violated control.

7.8.7   ALTER

This keyword alters the first control value for the well, defined by OPERATE (see 6.3.16), Second and other controls defined by OPERATE (see 6.3.16) can't be altered using ALTER.

The data should be entered in the following format:

ALTER well-name (well-number) value

where:
well-name (well-number) - well name (or number), whose first control value will be altered, value - new control value.

The keyword has an Eclipse compatible analogue WELTARG (see 6.10.17).

Example
```
DATE 1975 6 28
PRODUCER 7
OPERATE MAX STL 42 CONT
OPERATE MIN BHP 250 CONT
...
DATE 1975 9 30
ALTER 7 47.5
```

June 28, 1975 this example specifies for producer number 7 two controls: liquid rate (maximum 42) and bottom hole pressure (minimum 250). September 30, 1975 the value of liquid rate control was altered: new value is 47.5 (bottom hole pressure value stays the same).

7.8.8 GEOMETRY

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

The keyword specifies several geometric characteristics of the well. The data is used to obtain the well flow index.

The data should be entered in the following format:

GEOMETRY I (or J, or K) rad geofac wfrac skin

where:

- I (or J, or K) – one should enter the coordinate axis which is parallel to the wellbore (I – X axis, J – Y axis, K – Z axis);

- rad – well radius;

- geofac – geometric well factor;

- wfrac – a real number between 0 and 1, which specifies the part of circle which corresponds to this well. Usually a hole circle corresponds to the well (the well is inside the model) – 1. If the well is situated at the corner of the grid block on the grid boundary – 0.25. If the well is situated at the edge of the grid block on the grid boundary – 0.5;

- skin – skin.

The keyword has an Eclipse compatible analogue COMPDAT (see 6.10.6) (several parameters of COMPDAT correspond to GEOMETRY).

Example
```
WELL 8
PRODUCER 8
OPERATE MIN BHP 250 CONT
GEOMETRY K 0.0762 0.37 1. 0.
```

This example specifies for producer number 8 bottom hole pressure control (minimum 250). Using GEOMETRY are specified the following geometric characteristics of the well:

- the wellbore is parallel to the Z axis;

- well radius is equal to 0.0762 m;

- geometric well factor – 0.37;

- the hole circle is corresponding to the well;

- skin – 0.

7.8.9 PERF

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	✓ **CMG imex**	☐ **CMG stars**

The keywords sets grid blocks in which wellbore is situated. At first the well should be initialized using keywords WELL (see 7.8.3), PRODUCER (see 7.8.4), GEOMETRY (see 7.8.8).

The data should be entered in the following format:

PERF GEO wn location well-index (status) (connection)

where:

1. GEO – the keyword sets that the well index is calculated from the geometric information (last GEOMETRY (see 7.8.8)), dimensions and permeability of the grid blocks in which wellbore is situated.

2. wn – well number.

3. location – three numbers i, j, k – X, Y anf Z coordinates of grid block in which the wellbore is situated.

4. well-index – a number ff – multiplier. Well index is multiplied by this multiplier. (Well index is calculated from the geometric information (last GEOMETRY (see 7.8.8)), dimensions and permeability of the grid blocks in which wellbore is situated.)

5. status – block status: OPEN or CLOSED. OPEN – perforated interval is open (this is default status if status isn't specified). CLOSED – this block is specified to define the well trajectory. Perforated interval can be opened using the keyword PERF next time.

6. connection – this parameter sets the number of previous block (specified for this well using PERF keyword), i.e. the number of block in the direction of the flow at the time of production.
   The data should be entered in the following format: FLOW-TO ily.
   Where ily - previous block number of the word SURFACE, if the first (top) well block is specified.
   One can add here REFLAYER. This word marks the block where bottom hole pressure (BHP) is calculated. REFLAYER can be used only once for current PERF. If REFLAYER is not specified, THP is calculated for the first entered block.

The keyword has an Eclipse compatible analogue COMPDAT (see 6.10.6) (several parameters of COMPDAT correspond to PERF).

Example
```
PERF GEO 12
48 7 3 5. OPEN FLOW-TO 'SURFACE' REFLAYER
48 7 4 5. OPEN FLOW-TO 1
48 7 5 5. OPEN FLOW-TO 2
48 7 6 5. CLOSED FLOW-TO 3
48 7 7 5. OPEN FLOW-TO 4
```

In this example the word GEO sets that the well index is calculated from the geometric information (last GEOMETRY (see 7.8.8)), dimensions and permeability of the grid blocks in which wellbore is situated.

Next we specify five grid blocks in which the wellbore of well 12 is situated (1-st block (48, 7, 3), 2-nd block: (48, 7, 4), 3-rd block: (48, 7, 5), 4-th block: (48, 7, 6), 5-th block: (48, 7, 7)).

The multiplier ff is equal to 5 for all these blocks (well index is multiplied by this multiplier).

Perforated intervals are opened in all blocks except 4-th block. The flow direction from block to block is: 5-4-3-2-1-SURFACE. BHP is calculated for the 1-st block (the word REFLAYER is present for the 1-st block).

7.8.10　LAYERXYZ

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	☐	**CMG stars**

The keyword defines perforations which are situated on the deviated wellbore. Previously these perforations should be initialized using the keyword PERF (see 7.8.9).

The data should be entered in the following format:

LAYERXYZ wn location x1 y1 z1 x2 y2 z2 plength

where:

1. wn – well name (number).

2. location – three numbers i, j, k – X, Y anf Z coordinates of grid block in which the wellbore is situated. The part of wellbore in this block is considered as deviated.

3. x1 y1 z1 – Cartesian coordinates of the "entry point" for the deviated wellbore in this block. Points (x1, y1, z1) and (x2, y2, z2) define the wellbore direction.

4. x2 y2 z2 – Cartesian coordinates of the "exit point" for the deviated wellbore in this block. Points (x1, y1, z1) and (x2, y2, z2) define the wellbore direction.

5. plength – length of the perforated interval within the grid block. This length can be greated than the distance between (x1, y1, z1) and (x2, y2, z2).

The deviated well index in imex and stars:

$$\text{Well index} = \frac{2\pi \cdot wfrac \cdot K \cdot welllength \cdot ff}{\ln\left(r_{eff}/r_{well}\right) + skin},$$

where

- $wfrac$ $(0 \leqslant wfrac \leqslant 1)$ — well angular fraction;

- K — average permeability (the description of calculation is below);

- $welllength$ — length of the perforated interval within the grid block;

- ff — well index multiplier;

- r_{eff} — drainage radius (the description of calculation is below);

- r_{well} — well radius;

- $skin$ — skin.

Let $u = (u_x, u_y, u_z)$ — a unit vector in the wellbore direction; e_x, e_y, e_z — unit vectors (parallel to the vectors which joins centers of mass of the opposite edges in this block).

When u is parallel to one of the vectors e_x, e_y or e_z then

$$r_{eff}^{(j)} = r_{eff}(e_j) = geofac \cdot \sqrt{\frac{V}{\pi \cdot h(j) \cdot wfrac}} \tag{7.1}$$

где

- $j \in x, y, z$; $geofac$ — geometric well factor (see GEOMETRY (see 7.8.8));

- V — volume of the perforeted block; $h(j)$ — the grid block thikness in the direction j.

Let

$$\begin{aligned}
\alpha_x &= (u, e_x)^2 \cdot \left(1 - (u, e_y)^2\right) \cdot \left(1 - (u, e_z)^2\right) \\
\alpha_y &= \cdot(u, e_y)^2 \cdot \left(1 - (u, e_x)^2\right) \cdot \left(1 - (u, e_z)^2\right) \\
\alpha_z &= \cdot(u, e_z)^2 \cdot \left(1 - (u, e_x)^2\right) \cdot \left(1 - (u, e_y)^2\right)
\end{aligned}$$

In the general case the drainage radius is equil to:

$$r_{eff}(u) = \left(r_{eff}^{(x)} \cdot \alpha_x + r_{eff}^{(y)} \cdot \alpha_y + r_{eff}^{(z)} \cdot \alpha_z\right) / (\alpha_x + \alpha_y + \alpha_z) \tag{7.2}$$

tNavigator uses right formula:

$$r_{eff}(u) = \left(\left(r_{eff}^{(x)}\right)^2 \cdot \alpha_x + \left(r_{eff}^{(y)}\right)^2 \cdot \alpha_y + \left(r_{eff}^{(z)}\right)^2 \cdot \alpha_z\right)^{1/2} / (\alpha_x + \alpha_y + \alpha_z)^{1/2} \tag{7.3}$$

Average permeability K is calculated the same way: one should replace $r_{eff}^{(j)} = r_{eff}(e_j)$ by

$$K(e_j) = \sqrt{\prod_{\substack{1 \leqslant i \leqslant 3 \\ i \neq j}} k_i}.$$

Example
```
LAYERXYZ 14
8 10 4 394247.20378 373635.68824 1371.69128 394247.30153
373637.60444 1377.65544 0.00129
8 10 5 394247.30153 373637.60444 1377.65544 394247.77237
373646.83405 1406.38252 30.17704
8 10 6 394247.77237 373646.83405 1406.38252 394248.08057
373652.27302 1423.68686 18.14215
8 10 7 394248.08057 373652.27302 1423.68686 394248.61406
373660.44384 1450.54380 28.07750
```

This example sets four grid blocks in which wellbore of well 14 is deviated.

7.8.11 ON-TIME

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	☐ CMG stars

The keyword sets well efficiency factor (the fraction of time during which a well works). This number should be between 0.001 and 1. If this number is less than 0.001 it is set equal to 0.001 during the simulation.

The data should be entered in the following format:

ON-TIME well-name
OTF-input

where:

- well-name – well name (should be entered immediatelly after ON-TIME, on the same line);

- OTF-input – well efficiency factor (the fraction of time during which a well works); (should be entered on the next line after the line with ON-TIME).

The keyword has an Eclipse compatible analogue WEFAC (see 6.10.22).

```
Example
ON-TIME A2
0.6429
ON-TIME B3
1.
```

This example specifies well efficiency factor for the well A2 equal to 0.6429, for the well B3 – 1.

7.8.12　STOP

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	√ CMG imex	☐ CMG stars

The keyword terminates the simulation. The data after STOP is ignored.
The keyword has an Eclipse compatible analogue END (see).

Example
```
DATE 1973 4 17
STOP
```

The simulation terminates on April 17, 1973.

8   Keywords compatible with tNavigator and CMG STARS

For best adaptation of user experience the keyword notations are chosen to be close as much as possible to the most common simulators:

- Eclipse (c) Schlumberger,

- IMEX (c) Computer Modelling Group Ltd,

- STARS (c) Computer Modelling Group Ltd,

- MORE (c) Roxar.

Eclipse compatible keywords are red. For example: TABDIMS (see   6.1.13). Index of Eclipse compatible keywords – 11.

CMG compatible keywords are pink. For example: TEMR (see   8.4.4). Index of CMG compatible keywords – 12.

MORE compatible keywords are green. For example: IDATe (see 9.1.5). Index of MORE compatible keywords – 14.

The keywords which RFD uses only in tNavigator are blue. For example: REACCONC (see 6.8.46). Index of RFD compatible keywords – 13.

In the description of each keyword there is a table where a simulator is ticked off if this keyword is compatible with this simulator.

tNavigator reads keyword notations of these simulators and converts them into its inner data notations.
This section describes all keywords which can be used in tNavigator:

- tNavigator keywords;

- CMG STARS keywords.

This description pointed out if there are parameters of the keyword which are ignored by tNavigator or which use is different from CMG.

For convenience keyword are grouped in several sections similar to CMG stars sections.

- Input/Output Control (8.1)

- Reservoir description (8.2)

- Other Reservoir Properties (8.3)

- Component properties (8.4)

- Rock-Fluid data (8.5)

- Initial conditions (8.6)

- Numerical methods control (8.7)

- Geomechanical Model (8.8)

- Well and recurrent data (8.9)

All keyword names are case insensitive. Values of parameters usually follow on the next string. If keyword value is not specified by user the simulator uses a default value if it is defined or an error message is issued.

8.1   Input/Output Control

8.2   Reservoir description

8.2.1 VOLMOD

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets a multiplier $\gamma(x,y,z)$ for each grid block. Geometric volume V_{geom} is multiplied by $\gamma(x,y,z)$. The same number of values shoud be specified as the number of grid blocks.

Bulk grid block volume – section 3.25, pore block volume – section 3.24.

The keyword has an Eclipse compatible analogue MULTPV (see 6.2.21).

Default: 1.

Example
```
VOLMOD
120*2 200*1 2500*3
```

In this example $\gamma(x,y,z)$ in 120 blocks is – 2, in 200 blocks – 1, in 2500 blocks – 3.

8.2.2   NETPAY

	tNavigator		eclipse 300		MORE Roxar
✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	✓	**CMG stars**

The keyword sets net to gross values $\psi(x, y, z)$ for each grid block. The same number of values shoud be specified as the number of grid blocks.

Net fo gross values are used in porosity cacculations 3.51.
The keyword has an Eclipse compatible analogue NTG (see 6.2.18).

Default: 1.

8.3   Other Reservoir Properties

8.3.1   ROCKTYPE

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword is used to define multiple rock regions.

This keyword sets the number of rock region the following rock properties are assigned to. Rock properties: PRPOR (see 8.3.4), CPOR (see 8.3.5), CTPOR (see 8.3.6), ROCKCP (see 8.3.3), THCONG (see 8.3.11), THCONO (see 8.3.10), THCONS (see 8.3.12), THCONR (see 8.3.8), THCONW (see 8.3.9), THCONMIX (see 8.3.13), HLOSST (see 8.3.14), HLOSSPROP (see 8.3.16), HLOSSTDIFF (see 8.3.15).

The keyword THTYPE (see 8.3.2) specifies for each grid block the number of rock region to which it belongs.

```
Example
ROCKTYPE 1
PRPOR 16450
CPOR 1.2e-6
CTPOR 0.00015
ROCKCP 2.3E6 0
THCONR 187000
THCONS 5.1E5
THCONW 5.12E4
THCONO 1.22E4
THCONG 4000
THCONMIX SIMPLE
ROCKTYPE 2
PRPOR 16450
CPOR 1.4e-8
CTPOR 0.00015
ROCKCP 2.3E6 0
THCONR 187000
THCONS 4.5E5
THCONW 5.35E4
THCONO 1.11E4
THCONG 4000
THCONMIX SIMPLE
```

In this example rock properties are specified for 2 rock regions.

8.3.2   THTYPE

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword is used to define multiple rock regions.

The keyword THTYPE (see 8.3.2) specifies for each grid block the number of rock region to which it belongs. Only the number of rock region that has been defined earlier via ROCKTYPE (see 8.3.1) is allowed.

Default: all grid blocks belong to one region.

The keyword has an Eclipse compatible analogue ROCKNUM (see 6.4.6).

Example
```
ROCKTYPE 1
...
ROCKTYPE 2
...
THTYPE CON 2
```

In this example all grid blocks belong to the 2-nd rock region.

8.3.3 ROCKCP

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets the coefficients CP_1, CP_2 of the rock enthalphy formula 3.47:

$$H_R(T) = (CP_1(T - T_{ref}) + \frac{1}{2}CP_2(T - T_{ref})^2)$$

T_{ref} is set via TEMR (see 8.4.4).

The following parameters are to be specified:

1. CP_1 $(J/m^3/°C)$,

2. CP_2.

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: $CP_1 = 2347kJ/m^3$, $CP_2 = 0$.

The keyword has an Eclipse compatible analogues HEATCR (see 6.8.9), HEATCRT (see 6.8.10).

In tNavigator the coefficients CP_1, CP_2 can be specified via the keyword HEATTCR (see 6.8.11).

```
Example
ROCKTYPE 1
ROCKCP 3204500 0
```

In this example the coefficients of the rock enthalphy formula are specified for one rock region.

8.3.4 PRPOR

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the reference pressure p_{ref} (kPa), which is used in porosity calculations 3.51:

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1+c_p(p-p_{ref})-c_T(T-T_{ref})+c_{pT}(p-p_{ref})(T-T_{ref}))$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: the reference pressure is equal to the pressure in the first active grid block.

The keyword is analogous to the 1-st parameter of Eclipse compatible keyword ROCK (see 6.5.12).

Example
```
PRPOR 16550
```

In this example the reference pressure is $16550\,kPa$.

8.3.5 CPOR

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the rock compressibility coefficient c_p ($1/kPa$), which is used in porosity calculations 3.51:

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref}))$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: 0.

The keyword is analogous to the 2-nd parameter of Eclipse compatible keyword ROCK (see 6.5.12).

Example
```
CPOR 1.3e-6
```

In this example the rock compressibility coefficient is 1.3e-6 $1/kPa$.

8.3.6   CTPOR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets the effective thermal expansion coefficient of the formation c_T $(1/C)$, which is used in porosity calculations 3.51:

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref}))$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: 0.

The keyword is analogous to the 1-st parameter of ROCKT (see 6.8.16), which is used in tNavigator.

Example
```
CTPOR 0.00012
```

In this example the effective thermal expansion coefficient is $0.00012\,1/C$.

8.3.7 CPTPOR

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword sets the pressure-temperature cross-term coefficient of the formation effective porosity c_{pT} $(1/kPa - C)$, which is used in porosity calculations 3.51:

$$\phi(p,T,x,y,z) = \psi(x,y,z)\phi(x,y,z)(1 + c_p(p - p_{ref}) - c_T(T - T_{ref}) + c_{pT}(p - p_{ref})(T - T_{ref}))$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: 0.

The keyword is analogous to the 2-nd parameter of ROCKT (see 6.8.16), which is used in tNavigator.

Example
```
CPTPOR 0.000042
```

In this example the pressure-temperature cross-term coefficient of the formation effective porosity is $0.000042\,1/kPa - C$.

8.3.8 THCONR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies the rock thermal conductivity k_R ($J/m/day/C$), which is used in the block thermal conductivity calculations 3.56 when THCONMIX (see 8.3.13) sets the option SIMPLE:

$$K_b = \phi \left(1 - \widehat{S}_S\right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: $149.6\,kJ/m/day/C$.

The keyword has an Eclipse compatible analogue THCONR (see 6.8.14).
The keyword is analogous to the 1-st parameter THCONT (see 6.8.17), which is used in tNavigator.

Example
```
THCONR 187000
```

In this example the rock thermal conductivity is $187000\,J/m/day/C$.

8.3.9   THCONW

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies the water thermal conductivity k_W ($J/m/day/C$), which is used in the block thermal conductivity calculations 3.56 when THCONMIX (see 8.3.13) sets the option SIMPLE:

$$K_b = \phi \left(1 - \widehat{S}_S \right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword is analogous to the 2-nd parameter THCONT (see 6.8.17), which is used in tNavigator.

Example
`THCONW 4.85E4`

In this example the water thermal conductivity is $4.85E4\,J/m/day/C$.

8.3.10 THCONO

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword specifies the oil thermal conductivity k_O ($J/m/day/C$), which is used in the block thermal conductivity calculations 3.56 when THCONMIX (see 8.3.13) sets the option SIMPLE:

$$K_b = \phi \left(1 - \widehat{S}_S\right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword is analogous to the 3-rd parameter THCONT (see 6.8.17), which is used in tNavigator.

Example
THCONO 2.03E4

In this example the oil thermal conductivity is $2.03E4 J/m/day/C$.

8.3.11 THCONG

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies the gas thermal conductivity k_G ($J/m/day/C$), which is used in the block thermal conductivity calculations 3.56 when THCONMIX (see 8.3.13) sets the option SIMPLE:

$$K_b = \phi \left(1 - \widehat{S}_S \right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword is analogous to the 4-th parameter THCONT (see 6.8.17), which is used in tNavigator.

Example
```
THCONG 3800
```

In this example the gas thermal conductivity is $3800 J/m/day/C$.

8.3.12 THCONS

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies the solid phase thermal conductivity k_S ($J/m/day/C$), which is used in the block thermal conductivity calculations 3.56 when THCONMIX (see 8.3.13) sets the option SIMPLE:

$$K_b = \phi \left(1 - \widehat{S}_S\right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword is analogous to the 5-th parameter THCONT (see 6.8.17), which is used in tNavigator.

Example
```
THCONS 152000
```

In this example the solid phase thermal conductivity is $152000\,J/m/day/C$.

8.3.13 THCONMIX

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

The keyword sets the method of block thermal conductivity calculation: SIMPLE or COMPLEX.

The keyword has an analogue THCONMIX (see 6.8.18), which is used in tNavigator.

If the keyword THCONMIX specifies the option SIMPLE then **thermal conductivity of the grid block** is (3.56):

$$K_b = \phi \left(1 - \widehat{S}_S\right) \cdot (k_W S_W + k_O S_O + k_G S_G) + \phi \cdot k_S \cdot \widehat{S}_S + (1 - \phi) \cdot k_R$$

where

- k_P, $P = W, O, G, S$ – phase thermal conductivity (THCONW (see 8.3.9), THCONO (see 8.3.10), THCONG (see 8.3.11), THCONS (see 8.3.12)) (default THCONS (see 8.3.12) = THCONR (see 8.3.8));

- S_P, $P = W, O, G$ – phase saturation, $\widehat{S}_S$ – solid phase saturation;

- k_R – rock thermal conductivity (THCONR (see 8.3.8)) (default $149.6 kJ/m/day/C$);

- ϕ – porosity.

tNavigator also uses the keyword THCONT (see 6.8.17) to specify the parameters k_P, $P = W, O, G, S$ и k_R.

If the keyword THCONMIX specifies the option COMPLEX then **thermal conductivity of the grid block** is (3.56):

$$K_b' = \left(1 - \sqrt{S_W + S_O}\right) \cdot k_G \cdot F\left(\frac{k_R}{k_G}\right) + \sqrt{S_W + S_O} \cdot k_L \cdot F\left(\frac{k_R}{k_L}\right)$$

where

$$F(x) = \exp\left(\left(0.28 - 0.32876 \cdot \log \phi_f - 0.024755 \cdot \log x\right) \log x\right), \quad K_L = \frac{k_W S_W + K_O S_O}{S_W + S_O}$$

where

- ϕ_f – "mobile" porosity.

The dependence between **the block thermal conductivity** and the temperature

$$K_b = K_b' - 1.7524 \cdot 10^{-5}(T - T_{ref}) \cdot (K_b' - 119616)$$
$$\cdot \left(K_b'\right)^{-0.64} \cdot \left(K_b' \cdot \left(1.8 \cdot 10^{-3} \cdot T\right)^{\left(-3.6784 \cdot 10^{-6} K_b'\right)} + 110644.8\right)$$

where T_{ref} is given by the keyword TEMR (see 8.4.4).

In e300 data format **thermal conductivity of the grid block** is

$$K_b = (1 - \alpha S_G) \cdot k_R$$

where

- k_R – rock thermal conductivity (THCONR (see 6.8.14)) ($kJ/m/day/°C$);

- α – is set via THCONSF (see 6.8.15), $\alpha \in [0, 1]$ (default: 0);

- S_G – gas saturation.

Example
```
THCONMIX SIMPLE
```

This example specifies the method of block thermal conductivity calculation – SIMPLE.

8.3.14 HLOSST

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

This keyword specifies the initial temperature of reservoir surroundings ($°C$), (see the section "The heat exchange between the reservoir and surroundings" 3.29).

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword is analogous to the 2-nd parameter of Eclipse compatible keyword ROCKPROP (see 6.2.57).

Example
```
HLOSST 43
```

In this example the initial temperature of reservoir surroundings is $43°C$.

8.3.15 HLOSSTDIFF

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

This keyword specifies the minimal difference between temperatures when the calculations of the heat exchange should start ($^{\circ}$C), (see the section "The heat exchange between the reservoir and surroundings" 3.29).

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

Default: 0.

Example
```
HLOSSTDIFF 1
```

In this example the minimal difference between temperatures when the calculations of the heat exchange should start 1°C.

8.3.16 HLOSSPROP

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword sets the connection between the reservoir and cap and base rocks, volumetric heat capacity ($J/m^3 - C$) and rock conductivity ($J/m - day - C$) – 3.29.

The following parameters are to be specified:

1. OVERBUR or (and) UNDERBUR or (and) +I, -I, +J, -J, +K, -K. This parameters are not obligatory. One may not specify them or can specify part of them.

   - OVERBUR heat loss properties are applied to the outer grid block faces at the reservoir top;

   - UNDERBUR heat loss properties are applied to the outer grid block faces at the reservoir bottom;

   - +I, -I, +J, -J, +K, -K. Indicates the direction in which heat loss properties are applied (I – X-axis, J – Y-axis, K – Z-axis). OVERBUR and UNDERBUR can be used together with +I, -I, +J, -J.

2. volumetric heat capacity ($J/m^3 - C$);

3. rock conductivity ($J/m - day - C$).

Different values can be entered for different rock regions (see an example in the description of ROCKTYPE (see 8.3.1)).

The keyword has an Eclipse compatible analogues ROCKCON (see 6.2.58), ROCKPROP (see 6.2.57).
The keyword is also analogous to ROCKCONT (see 6.2.59), which is used in tNavigator.

Example
```
HLOSSPROP OVERBUR 1.7E6 1.002E5
```

In this example heat loss properties are applied to the outer grid block faces at the reservoir top, volumetric heat capacity is 1.7E6 $J/m^3 - C$, rock conductivity 1.002E5 $J/m - day - C$.

8.4   Component properties

8.4.1 MODEL

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the number of components and component volatility type.

The data should be entered in the following format:
MODEL

1. total number of components in the model (including water);

2. total number of components in the water, (or) oil, (or) gas phases;

3. total number of components in the water and (or) oil phases;

4. the number of water-like or aqueous components (default: 1 – water).

The keyword has an Eclipse compatible analogue COMPS (see 6.7.3). COMPS (see 6.7.3) sets the number of hydrocarbon components, MODEL – total number of components (including water).

In the example below there are 7 components. "1" indicates in the table that the component can be in this phase. There are 2 water-like components, 4 components – water-like or in the oil phase, 6 components – water-like, in the oil or gas phases. Solid phase (coke) is also enable.

Component name	Water	Oil	Gas	Solid phase
Water	1	0	1	0
Асфальтены	0	1	0	0
Light oil	0	1	1	0
CO_2	0	1	1	0
N_2 / CO	0	0	1	0
Oxygen	0	0	1	0
Coke	0	0	0	1

Example
```
MODEL 7 6 4 1
COMPNAME 'Water' 'Asphaltenes' 'Light Oil' 'CO2' 'N2CO'
'Oxygen' 'Coke'
```

8.4.2   COMPNAME

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

The keyword specifies component names. This keyword is used after the keyword MODEL (see 8.4.1)

COMPNAME should be followed by the component names.

The keyword has an Eclipse compatible analogue CNAMES (see 6.7.4).

In the example below there are 7 components. "1" indicates in the table that the component can be in this phase. There are 2 water-like components, 4 components – water-like or in the oil phase, 6 components – water-like, in the oil or gas phases. Solid phase (coke) is also enable.

Component name	Water	Oil	Gas	Solid phase
Water	1	0	1	0
Asphaltenes	0	1	0	0
Light oil	0	1	1	0
CO_2	0	1	1	0
N_2 / CO	0	0	1	0
Oxygen	0	0	1	0
Coke	0	0	0	1

Example
```
MODEL 7 6 4 1
COMPNAME 'Water' 'Asphaltenes' 'Light Oil' 'CO2' 'N2CO'
'Oxygen' 'Coke'
```

8.4.3   PRSR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

This keyword specifies the reference pressure. The reference pressure corresponds to the densities entered via the keywords MOLDEN (see 8.4.5), MASSDEN (see 8.4.6), SOLID_DEN.

The reference pressure is used in the formulas: 3.7, 3.9 and 3.17 (component phase density calculations).

The keyword has an Eclipse compatible analogues PREF (see 6.8.28), SPREF (see 6.8.22).

Example
PRSR 101

In this example reference pressure is equal to 101.

8.4.4 TEMR

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

This keyword specifies the reference temperature. The reference temperature corresponds to the densities entered via the keywords MOLDEN (see 8.4.5), MASSDEN (see 8.4.6), SOLID_DEN.

The reference temperature is used in the formulas: 3.7, 3.9, 3.17 (component phase density calculations), 3.38, 3.41, 3.46, 3.47, 3.51.

The keyword has an Eclipse compatible analogues TREF (see 6.8.30), STREF (see 6.8.24).

Example
```
TEMR 70
```

In this example reference temperature is equal to 101.

8.4.5 MOLDEN

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets the component molar density (mol/m^3) at reference temperature TEMR (see 8.4.4) and reference pressure PRSR (see 8.4.3).
The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The component molar density is used in the formulas 3.7, 3.9.
Component mass density (MASSDEN (see 8.4.6)) is equal to the product of the component molar density and molecular mass.

8.4.6 MASSDEN

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

The keyword sets the component mass density (kg/m^3) at reference temperature TEMR (see 8.4.4) and reference pressure PRSR (see 8.4.3).
The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The component mass density is used in the formulas 3.7, 3.9.
Component mass density is equal to the product of the component molar density (MOLDEN (see 8.4.5)) and molecular mass.

Example
```
MODEL 3 3 3 1
MASSDEN
982.12 964.17 267.25
```

This example sets the component mass density for 3 components (the number of components in the oil and water phases – 3-rd parameter of MODEL (see 8.4.1)).

8.4.7 CP

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the component compressibility ($1/kPa$), which is used in water mass and molar density calculations 3.7 and oil mass and molar density calculations 3.9.

The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue CREF (see 6.8.29).

Example
```
MODEL 3 3 3 1
CP
6.28e-007 3.7e-006 3.7e-006
```

This example sets the component compressibility for 3 components (the number of components in the oil and water phases – 3-rd parameter of MODEL (see 8.4.1)).

8.4.8   CT1

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets the first thermal expansion coefficient $(1/C)$ for each component. This coefficient is used in water mass and molar density calculations 3.7 and oil mass and molar density calculations 3.9.

The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue THERMEX1 (see 6.8.25).

Example
```
MODEL 3 3 3 1
CT1
0.0006643 4.8977e-006 4.3512e-006
```

This example sets the first thermal expansion coefficient for 3 components (the number of components in the oil and water phases – 3-rd parameter of MODEL (see 8.4.1)).

8.4.9　CT2

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the second thermal expansion coefficient ($1/C^2$) for each component. This coefficient is used in water mass and molar density calculations 3.7 and oil mass and molar density calculations 3.9.

Total thermal expansion coefficient is equal to $c_{k,1,T} + T * c_{k,2,T}$ The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an analogue THERMEX2 (see 6.8.26), which is used in tNavigator. There are no Eclipse compatible analogues.

Example
```
MODEL 3 3 3 1
CT2
0.00006643 4.8977e-007 4.3512e-007
```

This example sets the second thermal expansion coefficient for 3 components (the number of components in the oil and water phases – 3-rd parameter of MODEL (see 8.4.1)).

8.4.10 CPT

☑ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☑ CMG stars

The keyword sets the coefficient of density dependence on temperature and pressure ($1/kPaC$) for each component. This coefficient is used in water mass and molar density calculations 3.7 and oil mass and molar The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an analogue THERMEX3 (see 6.8.27), which is used in tNavigator. There are no Eclipse compatible analogues.

Example
```
MODEL 3 3 3 1
CPT
0.000006643 4.8977e-006 4.3512e-006
```

This example sets the coefficient of density dependence on temperature and pressure for 3 components (the number of components in the oil and water phases – 3-rd parameter of MODEL (see 8.4.1)).

8.4.11 PCRIT

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	√ CMG stars

The keyword sets the component critical pressure (kPa) which is used in gas mass and gas molar density calculations 3.14.

The same number of values as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue PCRIT (see 6.7.12). The difference is that Eclipse compatible PCRIT doesn't contain water critical pressure.

```
Example
MODEL 3 3 3 1
PCRIT
0 0 4326
```

This example sets the component critical pressure for 3 components (the number of components in the oil, water or gas phases – 2-nd parameter of MODEL (see 8.4.1)).

8.4.12 TCRIT

√	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	√	CMG stars

The keyword sets the component critical temperature (C) which is used in gas mass and gas molar density calculations 3.14.
The same number of values as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue TCRIT (see 6.7.11). The difference is that Eclipse compatible TCRIT doesn't contain water critical temperature.

```
Example
MODEL 3 3 3 1
TCRIT
0 0 -79.22
```

This example sets the component critical temperature for 3 components (the number of components in the oil, water or gas phases – 2-nd parameter of MODEL (see 8.4.1)).

8.4.13 SOLID_DEN

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the properties of component k in the solid phase that are used in the molar density calculations 3.17.

For each component the data should be entered in the following format:

1. 'component name' k (the number of component and component names are set via MODEL (see 8.4.1), COMPNAME (see 8.4.2));

2. density of component k (kg/m^3) at reference pressure PRSR (see 8.4.3) and reference temperature TEMR (see 8.4.4);

3. $c_{k,p}$ – compressibility of the component k ($1/kPa$);

4. $c_{k,T}$ – thermal expansion coefficient for component k ($1/C$);

5. $c_{k,pT}$ – the coefficient of density dependence on temperature and pressure ($1/kPa - C$).

The keyword has Eclipse compatible analogues SDREF (see 6.8.21), SCREF (see 6.8.23), STHERMX1 (see 6.8.19). For the parameter $c_{k,pT}$ tNavigator uses this keyword STHERMX2 (see 6.8.20).

8.4.14 SOLID_CP

	tNavigator		eclipse 300		MORE Roxar
✓		☐		☐	
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword sets the properties of component k in the solid phase that are used in the solid phase enthalpy calculations 3.46.

For each component the data should be entered in the following format:

1. 'component name' k (the number of component and component names are set via MODEL (see 8.4.1), COMPNAME (see 8.4.2));

2. coefficient $CP_{1,c}$ ($J/gmol - C$);

3. coefficient $CP_{2,c}$ ($J/gmol - C^2$);

This keyword is analoguos to SPECHS (see 6.8.62), SPECHT (see 6.8.63) which are used in tNavigator.

Example
```
SOLID_CP
'Comp1' 0.52 0.0076
```

In this example $CP_{1,c}$ и $CP_{2,c}$ are specified for the component Comp1.

8.4.15   KVTABLIM

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets the pressure and temperature range for $K_i(p,T)$ table. ($K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases.) These tables are used in the formulas of thermodynamic equilibrium condition 3.9.
The table $K_i(p,T)$ is set via KVTABLE (see 8.4.16).

The following parameters are to be specified:

1. The minimal value of pressure for $K_i(p,T)$ (kPa) – $plow$.

2. The maximal value of pressure for $K_i(p,T)$ (kPa) – $phight$.

3. The minimal value of temperature for $K_i(p,T)$ ($°C$) – $Tlow$.

4. The maximal value of temperature for $K_i(p,T)$ ($°C$) – $Thight$.

The keyword has an Eclipse compatible analogues KVTEMP (see 6.8.5), KVTABTn (see 6.8.6). tNavigator also uses the keyword KVTABLIM (see 6.8.7).

An example of use of KVTABLIM (see 8.4.15) is given in the description of the keyword KVTABLE (see 8.4.16).

8.4.16 KVTABLE

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the table $K_i(p,T)$. ($K_i(p,T)$ – the ratio of mole fractions of component in vapor and liquid phases.) These tables are used in the formulas of thermodynamic equilibrium condition 3.9.
The pressure and temperature range for $K_i(p,T)$ table is set via KVTABLIM (see 8.4.15).

The keyword has an Eclipse compatible analogues KVTEMP (see 6.8.5), KVTABTn (see 6.8.6). tNavigator also uses the keyword KVTABLIM (see 6.8.7).

After KVTABLE one should specify the component k name in quotes ' ' (the number of components and component names are set via MODEL (see 8.4.1), COMPNAME (see 8.4.2)) and the table for this component:

$$K(Tlow, plow) \quad \ldots \quad K(Tlow, phight)$$
$$\ldots \quad \ldots \quad \ldots$$
$$K(Thight, plow) \quad \ldots \quad K(Thight, phight)$$

Interpolation between table entries:

- between two $K_i(p,T)$ for two adjacent pressures: $K_i(p,T)$ varies linearly with the coefficient $1/p$;

- between two non-zero $K_i(p,T)$ for two adjacent temperatures: $ln(K_i(p,T))$ varies linearly with the coefficient $1/T$;

- between two $K_i(p,T)$ (one of the value entries is zero) for two adjacent temperatures: $K_i(p,T)$ varies linearly with the coefficient $1/T$.

Example
```
KVTABLIM
10 500 20 250
KVTABLE
'COMPONENT2'
0.0001 0.0008
0.01 0.09
```

In this example the $K_i(p,T)$ table for COMPONENT2 has 2 rows and 2 columns. The pressure varies within: 10 - 300 kPa, temperature varies within: 20 - 250 $^{\circ}C$.

8.4.17   KV1 / KV2 / KV3 / KV4 / KV5

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

These keywords specify the coefficients of the correlation formula (3.22):

$$K_i(p,T) = (A_i + B_i/p + C_i p) \cdot e^{-D_i/(T-E_i)} \tag{8.1}$$

where A_i – KV3, B_i – KV1, C_i – KV2, $-D_i$ – KV4, E_i – KV5.

After each keyword (KV1, ... , KV5) one should specify the same number of coefficients as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keywords have an Eclipse compatible analogue KVCR (see 6.8.4). 1-st parameter of KVCR (see 6.8.4) corresponds to KV3, 2-nd parameter – KV1, 3-rd parameter – KV2, 4-th parameter – KV4 (with opposite sign), 5-th parameter – KV5.

```
Example
MODEL 3 3 3 1
KV1
0 0 185967
KV2
0 0 -2.34122e-007
KV3
0 0 15.4327
KV4
0 0 -633.552
KV5
0 0 -321.88
```

In this example the coefficients of the correlation formula are specified for 3 components.

8.4.18   CPL1 / CPL2 / CPL3 / CPL4

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	√	**CMG stars**

These keywords specify the coefficients in the component liquid enthalphy calculations (3.39):

$$H_{c,O}(T) = \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficietns:
$CP_{1,c}$ – CPL1 $(kJ/kg/°C)$,
$CP_{2,c}$ – CPL2 $(kJ/kg/°C/°C)$,
$CP_{3,c}$ – CPL3 $(kJ/kg/°C/°C/°C)$,
$CP_{4,c}$ – CPL4 $(kJ/kg/°C/°C/°C/°C)$.
Default values: $CP_{1,c} = 0.5 Btu/lbmol/F = 0.5*1.05506/0.453592*1.8 kJ/mol/C = 2.0934 kJ/mol/C$,
the other coefficients – 0.

After each keyword CPL1, CPL2, CPL3, CPL4 one should specify the same number of coefficients as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

The keywords CPL1, CPL2 have an Eclipse compatible analogues SPECHA (see 6.8.53), SPECHB (see 6.8.54). tNavigator also uses the keywords $CP_{3,c}$ = SPECHC (see 6.8.55), $CP_{4,c}$ = SPECHD (see 6.8.56).

```
Example
CPL1
0.66 0.52
CPL2
0.0071 0.0054
CPL3
0.00062 0.00046
CPL4
0.000055 0.000078
```

In this example the coefficients of component liquid enthalphy formula are specified for 2 components.

8.4.19   CPG1 / CPG2 / CPG3 / CPG4

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

These keywords specify the coefficients in the component gas enthalphy calculations (3.43):

$$H_{c,G}(T) = h_{c,G} + \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients:
$h_{c,G}$ – HVAPR (see 8.4.20),
$CP_{1,c}$ – CPG1 $(kJ/kg/°C)$,
$CP_{2,c}$ – CPG2 $(kJ/kg/°C/°C)$,
$CP_{3,c}$ – CPG3 $(kJ/kg/°C/°C/°C)$,
$CP_{4,c}$ – CPG4 $(kJ/kg/°C/°C/°C/°C)$.
(CPG1 / CPG2 / CPG3 / CPG4 (see 8.4.19))

Default values: $h_{c,G} = 0.25 Btu/lb/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/kg/C = 1.0467 kJ/kg/C$, $CP_{1,c} = 0.25 Btu/lb/F = 1.0467 kJ/kg/C$, the other coefficients – 0.

After each keyword CPG1, CPG2, CPG3, CPG4 one should specify the same number of coefficients as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

The keywords CPG1, CPG2 have an Eclipse compatible analogues SPECHG (see 6.8.57), SPECHH (see 6.8.58). tNavigator also uses the keywords $CP_{3,c}$ = SPECHI (see 6.8.59), $CP_{4,c}$ = SPECHJ (see 6.8.60).

```
Example
CPG1
0.0066 0.0052
CPG2
0.00071 0.00054
CPG3
0.000062 0.000046
CPG4
0.0000055 0.0000078
```

In this example the coefficients of component gas enthalphy formula are specified for 2 components.

8.4.20 HVAPR

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

This keyword specifies the coefficients in the component gas enthalphy calculations (3.43):

$$H_{c,G}(T) = h_{c,G} + \sum_{i=1}^{4} \frac{1}{i} CP_{i,c}(T - T_{ref})^i$$

where the coefficients:
$h_{c,G}$ – HVAPR (see 8.4.20),
$CP_{1,c}$ – CPG1 ($kJ/kg/°C$),
$CP_{2,c}$ – CPG2 ($kJ/kg/°C/°C$),
$CP_{3,c}$ – CPG3 ($kJ/kg/°C/°C/°C$),
$CP_{4,c}$ – CPG4 ($kJ/kg/°C/°C/°C/°C$).
(CPG1 / CPG2 / CPG3 / CPG4 (see 8.4.19))

Default values: $h_{c,G} = 0.25 Btu/lb/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/kg/C = 1.0467 kJ/kg/C$, $CP_{1,c} = 0.25 Btu/lb/F = 1.0467 kJ/kg/C$, the other coefficients – 0.

After the keyword HVAPR one should specify the same number of coefficients as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue HEATVAPS (see 6.8.61).

Example
```
HVAPR 3*
```

In this example the coefficients $h_{c,G}$ of component gas enthalphy formula are specified on default for 3 components.

8.4.21 HVR

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

These keywords specify the coefficients in the vaporization enthalphy calculations (3.40):

$$HV_c(T) = A_c \cdot (1 - T/T_{c,crit})^{B_c} = A_c'(T_{c,crit} - T)^{B_c}, \quad A_c' = A_c/T_{c,crit}^{B_c}$$

where the coefficietns:
A_c' – HVR ($kJ/mol/C$),
B_c – EV (see 8.4.22).
Default values $A_c' = 0.25 Btu/lbmol/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/mol/C = 1.0467 kJ/mol/C$,
$B_c = 0.38$.
Critical temperature of the component $T_{c,crit}$ is specified via TCRIT (see 8.4.12).
If $HV_c(T) = 0$ then $T \geq T_{c,crit}$.

After each keyword HVR and EV (see 8.4.22) one should specify the same number of coefficients as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue HEATVAP (see 6.8.12).

Example
```
HVR
1.33 2.11
EV
0.42 0.39
```

In this example the coefficients of the vaporization enthalphy formula are specified for 2 components.

8.4.22 EV

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

These keywords specify the coefficients in the vaporization enthalphy calculations (3.40):

$$HV_c(T) = A_c \cdot (1 - T/T_{c,crit})^{B_c} = A'_c (T_{c,crit} - T)^{B_c}, \quad A'_c = A_c / T_{c,crit}^{B_c}$$

where the coefficietns:
A'_c – HVR (see 8.4.21) ($kJ/mol/C$),
B_c – EV.
Default values $A'_c = 0.25 Btu/lbmol/F = 0.25 * 1.05506/0.453592 * 1.8 kJ/mol/C = 1.0467 kJ/mol/C$,
$B_c = 0.38$.
Critical temperature of the component $T_{c,crit}$ is specified via TCRIT (see 8.4.12).
If $HV_c(T) = 0$ then $T \geq T_{c,crit}$.

After each keyword HVR (see 8.4.21) and EV one should specify the same number of coefficients as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

The keyword has an Eclipse compatible analogue HEATVAPE (see 6.8.13).

Example
HVR
1.33 2.11
EV
0.42 0.39

In this example the coefficients of the vaporization enthalphy formula are specified for 2 components.

8.4.23 STOREAC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword specifies the stoichiometric coefficients for all reactants in each chemical reaction as a line of numbers. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

Each line should contain the same number of values as the number of components (first parameter of MODEL (see 8.4.1)). If the component isn't the reactant of the reaction one should enter zero.

The keyword has an Eclipse compatible anologue STOREAC (see 6.8.49).

For the chemical reaction

$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$

and five components there is an example:

Example
```
STOPROD 0 0 0 3 4
STOREAC 0 1 5 0 0
```

The reactants of this reaction are: the 2nd component (with the coefficient 1) and the 3rd component (with the coefficient 5). 1st component isn't present in this reaction The products of this reaction are: the 4th component (with the coefficient 3) and the 5th component (with the coefficient 4).

8.4.24   STOPROD

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword specifies the stoichiometric coefficients for all products in each chemical reaction as a line of numbers. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

Each line should contain the same number of values as the number of components (first parameter of MODEL (see 8.4.1)). If the component isn't the product of the reaction one should enter zero.

The keyword has an Eclipse compatible anologue STOPROD (see 6.8.48).

For the chemical reaction
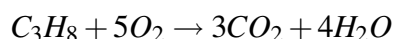
$$C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$$

and five components there is an example:

Example
```
STOPROD 0 0 0 3 4
STOREAC 0 1 5 0 0
```

The reactants of this reaction are: the 2nd component (with the coefficient 1) and the 3rd component (with the coefficient 5). 1st component isn't present in this reaction The products of this reaction are: the 4th component (with the coefficient 3) and the 5th component (with the coefficient 4).

8.4.25 FREQFAC

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the reaction rate A_r of each chemical reaction. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword has an Eclipse compatible anologue REACRATE (see 6.8.42).

Example
FREQFAC 0.0000038

In this example reaction rate is equal to 0.0000038.

8.4.26　EACT

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets the activation energy in chemical reaction rates E_r of each chemical reaction ($J/gmol$). Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword has an Eclipse compatible anologue REACACT (see 6.8.43).

Example
EACT 18400

In this example activation energy is equal to 18400 $J/gmol$.

8.4.27 RENTH

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the reaction enthalpy H_r of each chemical reaction ($J/gmol$). Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPRODCMG (see 6.8.48), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword has an Eclipse compatible anologue REACENTH (see 6.8.52).

Example
```
EACT 15200
```

In this example reaction enthalpy is equal to 15200 $J/gmol$.

8.4.28   RORDER

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies the order of component terms N_r of each chemical reaction as a line of numbers. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

Each line should contain the same number of values as the number of components (first parameter of MODEL (see 8.4.1)). If the reaction rate doesn't depend of the component concentration one should enter zero.

The keyword has an Eclipse compatible anologues REACCORD (see 6.8.44), REACSORD (see 6.8.51).

Example
RORDER 0 0 1 1 0 0

In this example there are 6 components. The rate of this reaction depends (linearly) of the concentration of 3rd and 4th components.

8.4.29   RPHASE

	tNavigator		eclipse 300		MORE Roxar
✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword sets the component phase in each chemical reaction. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

Each line should contain the same number of values as the number of components (first parameter of MODEL (see 8.4.1)).

The values could be the following:

- 0 component doesn't react;

- 1 component reacts in water phase;

- 2 component reacts in oil phase;

- 3 component reacts in gas phase;

- 4 component reacts in solid phase.

The keyword has an Eclipse compatible anologue REACPHA (see 6.8.50).

Example
```
RPHASE 0 0 2 3 3 4
```

In this example there are 6 components. 1-st and 2-nd component doesn't react, 3-rd component reacts in oil phase, 4-th and 5-th – gas phase, 6-th – solid phase.

8.4.30 RTEMUPR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets the maximal temperature T_u (C), which is used in the reaction rate calculations 3.57. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword has an Eclipse compatible anologue REACLIMS (see 6.8.45).

Example
```
RTEMUPR 230
```

In this example the maximal temperature is equal to $230°C$.

8.4.31 RTEMLOWR

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the minimal temperature T_l (C), which is used in the reaction rate calculations 3.57. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword has an Eclipse compatible anologue REACLIMS (see 6.8.45).

Example
```
RTEMLOWR 120
```

In this example the minimal temperature is equal to $120°C$.

8.4.32   RXCRITCON

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets the critical value of reactant concentration (kPa, if the pressure is used or $gmol/m3$, if the concentration is used), which is used in the reaction rate calculations 3.57. Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The data should be entered in the followind format:

- 'component name';

- critical value of concentration.

The keyword has an anologue REACCONC (see 6.8.46), which is used in tNavigator. There are no Eclipse compatible analogues.

Example
```
RXCRITCON 'comp1' 0.00022
```

In this example the critical value of **comp1** concentration is equal to 0.00022.

8.4.33   O2PP

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets that the gas partial pressure will be used in calculations of c'_{ri} in the formula 3.59. The keyword O2PP is used only for components in gas phase (if the 'component name' is entered) and is the default value for oxygen (if the 'component name' is not entered). Full description of chemical reactions is in the section 3.28.

The number of reaction (for which all properties are specified) is not set explicitly. So for first reaction all keywords STOPROD (see 8.4.24), STOREAC (see 6.8.49), FREQFAC (see 8.4.25), RENTH (see 8.4.27) etc. should appear as a group. Then for second reaction all these keywords should apper, for third reaction, etc.

The keyword is analogous to GPP in Eclipse compatible keyword REACPHA (see 6.8.50).

Example
O2PP

In this example in calculations of c'_{ri} will be used oxygen partial pressure.

8.4.34 VSTYPE

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword is used to define multiple viscosity regions.

The keyword specifies for each grid block the number of viscosity region to which it belongs. Only the number of viscosity region that has been defined earlier via VISCTYPE (see 8.4.35) is allowed.

Default: all grid blocks belong to one region.

The keyword is analogous to VISCNUM (see 6.4.9), which is used in tNavigator.

```
Example
VISCTYPE 1
...
VISCTYPE 2
...
VSTYPE 2 2 2 2 2 1 1 1 1 1 1 1
2 2 2 2 1 1 1 1 1 1 1 1 1 1 1
```

There are two viscosity regions in this example.

8.4.35   VISCTYPE

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword is used to define multiple viscosity regions.

This keyword sets the number of viscosity region the following viscosity properties are assigned to. Viscosity data: AVISC (see 8.4.36), BVISC (see 8.4.37), VISCTABLE (see 8.4.38), VSMIXCOMP (see 8.4.39), VSMIXCOMP (see 8.4.39), VSMIXFUNC (see 8.4.41), AVG (see 8.4.42), BVG (see 8.4.43).

The keyword VSTYPE (see 8.4.34) specifies for each grid block the number of viscosity region to which it belongs.

```
Example
VISCTYPE 1
AVISC 0.4 0.5 0.6
BVISC 12 18 20
AVG 0.00022 0.00017
BVG 0.8 0.7
...
VISCTYPE 2
AVISC 0.3 0.51 0.62
BVISC 14 17 21
AVG 0.00022 0.00017
BVG 0.8 0.7
...
VSTYPE 2 2 2 2 2 1 1 1 1 1 1 1
2 2 2 2 1 1 1 1 1 1 1 1 1 1 1 1
```

There are two viscosity regions in this example.

8.4.36 AVISC

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

This keyword specifies coefficients in water and oil viscosity correlation formulas A_W (3.27) and A'_k (3.29).
B_W and B'_k are specified via BVISC (see 8.4.37).

The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

In e300 data format the coefficients of correlations are set via OILVISCC (see 6.8.37). Grabovski correlation is used in water viscosity calculations 3.26.

Example
```
AVISC 0.4 0.5 0.6
BVISC 12 18 20
```

In this example correlation coefficients are specified for 3 components.

8.4.37 BVISC

✓	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

This keyword specifies coefficients in water and oil viscosity correlation formulas B_W (3.27) and B'_k (3.29).
A_W and A'_k are specified via AVISC (see 8.4.36).

The same number of values as the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)) should be specified.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

In e300 data format the coefficients of correlations are set via OILVISCC (see 6.8.37). Grabovski correlation is used in water viscosity calculations 3.26.

Example
```
AVISC 0.4 0.5 0.6
BVISC 12 18 20
```

In this example correlation coefficients are specified for 3 components.

8.4.38   VISCTABLE

	tNavigator		eclipse 300		MORE Roxar
✓		☐		☐	
☐	eclipse 100	☐	CMG imex	✓	CMG stars

This keyword specifies the table viscosity-versus-temperature dependence (water viscosity – 3.27, oil viscosity 3.29).

One should enter a table. Each row of this table consists of parameters:

- temperature (C);

- viscosity for each component at this temperature (cp), (number of values should be equal to the number of components in the oil or water phases (3-rd parameter of MODEL (see 8.4.1)))

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

In e300 data format water viscosity as a function of temperature is specified via the keyword WATVISCT (see 6.8.35), oil viscosity – OILVISCT (see 6.8.36).

```
Example
VISCTABLE
7 1.45 116456 1199
49 0.77 720 9.2
115 0.31 19.4 0.19
221 0.16 1.12 0.02
309 0.08 0.12 0.01
```

In this example viscosity values are specified for 3 components at 5 temperatures.

8.4.39 VSMIXCOMP

☑ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	☑ CMG stars

This keyword specifies the function $f_k(x)$ (which is used in oil viscosity calculations – 3.29).

Function $f_k(x)$ should be specified via 3 keywords:

- VSMIXCOMP (see 8.4.39): 'component name';

- VSMIXENDP (see 8.4.40): minimal and maximum values of x (in a range from 0 to 1);

- VSMIXFUNC (see 8.4.41): 11 values of $f_k(x)$.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

The keyword has an Eclipse compatible analogue OILVINDX (see 6.8.38). tNavigator also uses the keyword OILVINDT (see 6.8.39).

```
Example
VSMIXCOMP 'Comp1'
VSMIXENDP 0 0.40
VSMIXFUNC 0 0.04 0.08 0.12 0.16 0.20 0.24 0.28 0.32 0.36 0.40
```

In this example $f_k(x)$ is specified for component Comp1.

8.4.40 VSMIXENDP

✓	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	✓	CMG stars

This keyword specifies the function $f_k(x)$ (which is used in oil viscosity calculations – 3.29).

Function $f_k(x)$ should be specified via 3 keywords:

- VSMIXCOMP (see 8.4.39): 'component name';

- VSMIXENDP (see 8.4.40): minimal and maximum values of x (in a range from 0 to 1);

- VSMIXFUNC (see 8.4.41): 11 values of $f_k(x)$.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

The keyword has an Eclipse compatible analogue OILVINDX (see 6.8.38). tNavigator also uses the keyword OILVINDT (see 6.8.39).

```
Example
VSMIXCOMP 'Comp1'
VSMIXENDP 0 0.40
VSMIXFUNC 0 0.04 0.08 0.12 0.16 0.20 0.24 0.28 0.32 0.36 0.40
```

In this example $f_k(x)$ is specified for component Comp1.

8.4.41 VSMIXFUNC

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

This keyword specifies the function $f_k(x)$ (which is used in oil viscosity calculations – 3.29).

Function $f_k(x)$ should be specified via 3 keywords:

- VSMIXCOMP (see 8.4.39): 'component name';

- VSMIXENDP (see 8.4.40): minimal and maximum values of x (in a range from 0 to 1);

- VSMIXFUNC (see 8.4.41): 11 values of $f_k(x)$.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

The keyword has an Eclipse compatible analogue OILVINDX (see 6.8.38). tNavigator also uses the keyword OILVINDT (see 6.8.39).

```
Example
VSMIXCOMP 'Comp1'
VSMIXENDP 0 0.40
VSMIXFUNC 0 0.04 0.08 0.12 0.16 0.20 0.24 0.28 0.32 0.36 0.40
```

In this example $f_k(x)$ is specified for component Comp1.

8.4.42   AVG

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

This keyword specifies coefficients A_k in gas viscosity correlation formulas 3.32. B_k are specified via BVG (see 8.4.43).

The same number of values as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

In e300 data format the coefficients of correlations are set via GASVISCF (see 6.8.41).

```
Example
AVG 0.00022 0.00017
BVG 0.8 0.7
```

In this example correlation coefficients are specified for 2 components.

8.4.43 BVG

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

This keyword specifies coefficients B_k in gas viscosity correlation formulas 3.32. A_k are specified via AVG (see 8.4.42).

The same number of values as the number of components in the oil, water or gas phases (2-nd parameter of MODEL (see 8.4.1)) should be specified.

Different values can be entered for different viscosity regions (see an example in the description of VISCTYPE (see 8.4.35)).

In e300 data format the coefficients of correlations are set via GASVISCF (see 6.8.41).

```
Example
AVG 0.00022 0.00017
BVG 0.8 0.7
```

In this example correlation coefficients are specified for 2 components.

8.4.44 CMM

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword specifies component molecular weight ($kg/gmol$), which is used in gas viscosity calculations 3.33

The same number of values as the number of components (1-st parameter of MODEL (see 8.4.1)) should be specified.

In e300 data format molecular weight of hydrocarbon components is specified via the keyword MW (see 6.7.14), molecular weight of water components – MWW.

Example
CMM
0.011 0.35 0.05614

In this example molecular weight of 3 components is specified.

8.5 Rock-Fluid data

8.5.1   KRTYPE

	tNavigator		eclipse 300		MORE Roxar
✓		☐		☐	
☐	eclipse 100	☐	CMG imex	✓	CMG stars

The keyword is the full analogue of RTYPE (see 7.5.5).

8.5.2   KRTEMTAB

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☑ **CMG stars**

The keyword specifies temperature dependence for critical saturations and endpoints. This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4). The data should be specified for each saturation table region RPT (see 7.5.2).

Data format:

KRTEMTAB keyword(1) … keyword(m)

Where:
keyword(1) … keyword(m) – the keywords from the following list: SWR (see 8.5.3), SWCRIT (see 8.5.5), SOIRW (see 8.5.7), SGCON (see 8.5.9), SGR (see 8.5.11), SOIRG (see 8.5.13), SORW (see 8.5.15), SORG (see 8.5.17), KRWIRO (see 8.5.19), KRGCW (see 8.5.21), KROCW (see 8.5.23), PCGEND (see 8.5.25), PCWEND (see 8.5.27).

Each table row consists of parameters:

- temperature ($^{\circ}$C);

- the value of keyword(1) at this temperature;

- ...

- the value of keyword(m) at this temperature.

The keyword has an Eclipse compatible analogue ENPTVT (see 6.8.65).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keywords in the list above (SWR (see 8.5.3), SWCRIT (see 8.5.5) etc), specify properties in whole saturation table region. To specify properties in each grid block, use the keywords BSWR (see 8.5.4), BSWCRIT (see 8.5.6), BSOIRW (see 8.5.8), BSGCON (see 8.5.10), BSGR (see 8.5.12), BSOIRG (see 8.5.14), BSORW (see 8.5.16), BSORG (see 8.5.18), BKRWIRO (see 8.5.20), BKRGCW (see 8.5.22), BKROCW (see 8.5.24), BPCGMAX (see 8.5.26), BPCWMAX (see 8.5.28).

Example
```
RPT 1
SWT
0.30 0 1 1
0.35 0.005 0.5 0.9
0.60 0.036 0.01 0.3
0.74 0.121 0 0.01
SLT
0.65 0.14 0 0.0
0.72 0.039 0.01 0.0
0.89 0.01 0.178 0.0
0.99 0 1 0.0
KRTEMTAB SORW SOIRW PCWEND
0 0.311 0.311 0
50 0.3 0.3 0
100 0.14 0.14 14
200 0.049 0.049 44
300 0.023 0.023 76
400 0.02 0.02 99
RPT 2
SWT
0.04 0 1 0
0.96 1 0 0
SLT
0.04 1 0.0 0.0
0.96 0.0 1 0.0
KRTEMTAB SORW SOIRW
0 0.04 0.04
50 0.04 0.04
100 0.04 0.04
200 0.04 0.04
300 0.04 0.04
400 0.04 0.04
```

In this example critical saturations and endpoints versus temperature are specified for two saturation point regions (regions are set using RPT (see 7.5.2)).

8.5.3   SWR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{WL} – minimal value of water saturation S_W for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 2-nd parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSWR (see 8.5.4).

8.5.4 BSWR

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets S_{WL} – minimal value of water saturation S_W in each grid block. The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SWL (see 6.6.14).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SWR (see 8.5.3).

8.5.5 SWCRIT

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{WCR} – maximal (critical) value of water saturation S_W (for which $k_{rW}(S_W) = 0$) for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 3-rd parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSWCRIT (see 8.5.6).

8.5.6 BSWCRIT

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{WCR} – maximal (critical) value of water saturation S_W (for which $k_{rW}(S_W) = 0$) in ecah grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SWCR (see 6.6.16).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SWCRIT (see 8.5.5).

8.5.7 SOIRW

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{WU} – maximal value of water saturation S_W for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 4-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSOIRW (see 8.5.8).

8.5.8   BSOIRW

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keywors sets S_{WU} – maximal value of water saturation S_W in each grid block. The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SWU (see 6.6.20).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SOIRW (see 8.5.7).

8.5.9 SGCON

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keywors sets S_{GL} – minimal value of gas saturation S_G for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 5-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSGCON (see 8.5.10).

8.5.10　BSGCON

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{GL} – minimal value of gas saturation S_G in each grid block. The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SGL (see 6.6.15).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SGCON (see 8.5.9).

8.5.11 SGR

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{GCR} – maximal (critical) value of gas saturation S_G (for which $k_{rG}(S_G) = 0$) for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 6-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSGR (see 8.5.12).

8.5.12   BSGR

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{GCR} – maximal (critical) value of gas saturation S_G (for which $k_{rG}(S_G) = 0$) in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SGCR (see 6.6.17).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SGR (see 8.5.11).

8.5.13　SOIRG

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{GU} – maximal value of gas saturation S_G for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 7-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSOIRG (see 8.5.14).

8.5.14 BSOIRG

√	tNavigator	☐	eclipse 300	☐	MORE Roxar
☐	eclipse 100	☐	CMG imex	√	CMG stars

The keyword sets S_{GU} – maximal value of gas saturation S_G in each grid block. The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SGU (see 6.6.21).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SOIRG (see 8.5.13).

8.5.15   SORW

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{OWCR} – maximal (critical) value of the function $S_O = 1 - S_W - S_{GL}$, for which $k_{rOW}(S_W) = 0$ for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 8-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSORW (see 8.5.16).

8.5.16 BSORW

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{OWCR} – maximal (critical) value of the function $S_O = 1 - S_W - S_{GL}$, for which $k_{rOW}(S_W) = 0$ in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SOWCR (see 6.6.18).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SORW (see 8.5.15).

8.5.17 SORG

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets S_{OGCR} – maximal (critical) value of the function $S_O = 1 - S_G - S_{WL}$, for which $k_{rOG}(S_G) = 0$ for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 9-th parameter of Eclipse compatible keyword ENPTVT (see 6.8.65).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BSORG (see 8.5.18).

8.5.18 BSORG

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets S_{OGCR} – maximal (critical) value of the function $S_O = 1 - S_G - S_{WL}$, for which $k_{rOG}(S_G) = 0$ in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue SOGCR (see 6.6.19).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword SORG (see 8.5.17).

8.5.19 KRWIRO

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets k_{rWmax} – maximal value of water relative permeability $k_{rW}(S_W)$ for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 2-nd parameter of Eclipse compatible keyword ENKRVT (see 6.8.66).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BKRWIRO (see 8.5.20).

8.5.20 BKRWIRO

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets $k_{rW\,max}$ – maximal value of water relative permeability $k_{rW}(S_W)$ in each grid block.

The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue KRW (see 6.6.26).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword KRWIRO (see 8.5.19).

8.5.21   KRGCW

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets k_{rGmax} – maximal value of gas relative permeability $k_{rG}(S_G)$ for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 3-rd parameter of Eclipse compatible keyword ENKRVT (see 6.8.66).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BKRGCW (see 8.5.22).

8.5.22 BKRGCW

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets k_{rGmax} – maximal value of gas relative permeability $k_{rG}(S_G)$ in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue KRG (see 6.6.27).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword KRGCW (see 8.5.21).

8.5.23   KROCW

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	✓	**CMG stars**

The keyword sets k_{rOmax} – maximal value of oil relative permeability (functions $k_{rOW}(S_W)$ and $k_{rOG}(S_G)$) for one saturation table region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 4-th parameter of Eclipse compatible keyword ENKRVT (see 6.8.66).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BKROCW (see 8.5.24).

8.5.24 BKROCW

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets k_{rOmax} – maximal value of oil relative permeability (functions $k_{rOW}(S_W)$ and $k_{rOG}(S_G)$) in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue KRO (see 6.6.25).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword KROCW (see 8.5.23).

8.5.25 PCGEND

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets P_{CGmax} – maximal value of gas capillary pressure $P_{cOG}(S_G)$ (*bars*) for one saturation point region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 2-nd parameter of Eclipse compatible keyword ENPCVT (see 6.8.67).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BPCGMAX (see 8.5.26).

8.5.26 BPCGMAX

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets P_{CGmax} – maximal value of gas capillary pressure $P_{cOG}(S_G)$ (*bars*) in each grid block.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue PCG (see 6.6.29).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword PCWEND (see 8.5.27). PCGEND (see 8.5.25).

8.5.27 PCWEND

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets P_{CWmax} – maximal value of water capillary pressure $P_{cOW}(S_W)$ (*bars*) for one saturation point region (RPT (see 7.5.2)).

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

The keyword is analoguous to the 3-rd parameter of Eclipse compatible keyword ENPCVT (see 6.8.67).

This keyword is used with KRTEMTAB (see 8.5.2); in the description of KRTEMTAB (see 8.5.2) there is an example of theirs usage.

To specify properties in each grid block (not in saturation point region), use the keyword BPCWMAX (see 8.5.28).

8.5.28 BPCWMAX

✓ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	✓ **CMG stars**

The keyword sets P_{CWmax} – maximal value of water capillary pressure $P_{cOW}(S_W)$ (*bars*) in each grid blocks.
The same number of values as the number of grid blocks should be entered.

Detailed description of phase relative permeabilities scaling is in the section 3.34.2.

This keyword has an Eclipse compatible analogue PCW (see 6.6.28).

This keyword should be entered after the keywords SWT (see 7.5.3), SLT (see 7.5.4), KRTEMTAB (see 8.5.2).

To specify properties in whole saturation table region (not in each grid block), use the keyword PCWEND (see 8.5.27).

8.6 Initial conditions

8.6.1 INITREGION

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets the initialization region number. The data below is assigned to this region. The following keywords can be specified after INITREGION: REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5).

INTYPE (see 8.6.2) for every grid block specifies the initialization region to which it belongs.

```
Example
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INITREGION 2
REFPRES 8500
REFDEPTH 1230
DWOC 1530
DGOC 1230
```

In this example there are two initialization regions. For each region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given.

8.6.2   INTYPE

√ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	√ CMG stars

The keyword should be followed by one integer for every grid block specifying the initialization region to which it belongs.

Initialization regions are specified using INITREGION (see 8.6.1), properties of each region – REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5).

```
Example
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INTYPE CON 1
```

In this example there is one initialization region. For this region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given. All grid blocks belong to this initialization region (CON specifies the constant value array).

8.6.3   REFPRES

√	**tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐	**eclipse 100**	√ **CMG imex**	√ **CMG stars**

The keyword specifies the reference pressure at the reference depth – REFDEPTH (see 8.6.4).
REFDEPTH and REFPRES can be specified for multiple initialization regions (use INITREGION (see 8.6.1)).

The keywords REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5), DGOC (see 8.6.6), VERTICAL (see 7.6.2) have an Eclipse compatible analogue EQUIL (see 6.9.1). 1-st parameter of EQUIL (see 6.9.1) corresponds to REFDEPTH (see 8.6.4), 2-nd – REFPRES (see 8.6.3), 3-rd – DWOC (see 8.6.5), 5-th – DGOC (see 8.6.6).

```
Example
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INITREGION 2
REFPRES 8500
REFDEPTH 1230
DWOC 1530
DGOC 1230
```

In this example there are two initialization regions. For each region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given.

8.6.4   REFDEPTH

☑ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☑ CMG imex	☑ CMG stars

The keyword specifies the reference depth. At the reference depth the keyword REFPRES (see 8.6.3) specifies the reference pressure.
REFDEPTH and REFPRES can be specified for multiple initialization regions (use INITREGION (see 8.6.1)).

The keywords REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5), DGOC (see 8.6.6), VERTICAL (see 7.6.2) have an Eclipse compatible analogue EQUIL (see 6.9.1). 1-st parameter of EQUIL (see 6.9.1) corresponds to REFDEPTH (see 8.6.4), 2-nd – REFPRES (see 8.6.3), 3-rd – DWOC (see 8.6.5), 5-th – DGOC (see 8.6.6).

Example
```
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INITREGION 2
REFPRES 8500
REFDEPTH 1230
DWOC 1530
DGOC 1230
```

In this example there are two initialization regions. For each region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given.

8.6.5 DWOC

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	✓	**CMG stars**

The keyword sets the water-oil contact depth.
DWOC can be specified for multiple initialization regions (use INITREGION (see 8.6.1)).

If VERTICAL DEPTH_AVE (VERTICAL (see 7.6.2)) is present, the resulting water saturation will reflect the water-oil transition zone caused by non-zero capillary pressure.

The keywords REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5), DGOC (see 8.6.6), VERTICAL (see 7.6.2) have an Eclipse compatible analogue EQUIL (see 6.9.1). 1-st parameter of EQUIL (see 6.9.1) corresponds to REFDEPTH (see 8.6.4), 2-nd – REFPRES (see 8.6.3), 3-rd – DWOC (see 8.6.5), 5-th – DGOC (see 8.6.6).

```
Example
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INITREGION 2
REFPRES 8500
REFDEPTH 1230
DWOC 1530
DGOC 1230
```

In this example there are two initialization regions. For each region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given.

8.6.6 DGOC

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	√	**CMG imex**	√	**CMG stars**

The keyword sets the gas-oil contact depth.
DGOC can be specified for multiple initialization regions (use INITREGION (see 8.6.1)).

If VERTICAL DEPTH_AVE (VERTICAL (see 7.6.2)) is present, the resulting gas saturation will reflect the liquid-gas transition zone caused by non-zero capillary pressure.

The keywords REFPRES (see 8.6.3), REFDEPTH (see 8.6.4), DWOC (see 8.6.5), DGOC (see 8.6.6), VERTICAL (see 7.6.2) have an Eclipse compatible analogue EQUIL (see 6.9.1). 1-st parameter of EQUIL (see 6.9.1) corresponds to REFDEPTH (see 8.6.4), 2-nd – REFPRES (see 8.6.3), 3-rd – DWOC (see 8.6.5), 5-th – DGOC (see 8.6.6).

```
Example
INITREGION 1
REFDEPTH 4500
REFPRES 1600
DWOC 2500
INITREGION 2
REFPRES 8500
REFDEPTH 1230
DWOC 1530
DGOC 1230
```

In this example there are two initialization regions. For each region reference depth (REFDEPTH (see 8.6.4)), reference pressure (REFPRES (see 8.6.3)), water-oil contact depth (DWOC (see 8.6.5)) and gas-oil contact depth (DGOC (see 8.6.6)) are given.

8.6.7 SO

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	✓ CMG stars

The keyword sets the initial oil saturation. The same number of values as the number of grid blocks should be specified. The values of oil saturation should be in the range from 0 to 1.

The keyword has an Eclipse compatible analogue SOIL (see 6.9.11).

Example
```
SO CON 0.7
```

In this example the initial oil saturation in all grid blocks is equal to 0.7 (CON (see 7.1.3) specifies the constant value array).

8.6.8   SG

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	☐ CMG imex	✓ CMG stars

The keyword sets the initial gas saturation. The same number of values as the number of grid blocks should be specified. The values of gas saturation should be in the range from 0 to 1.

The keyword has an Eclipse compatible analogue SGAS (see 6.9.10).

Example
```
SG CON 0
```

In this example the initial gas saturation in all grid blocks is equal to 0 (CON (see 7.1.3) specifies the constant value array).

8.6.9 SW

✓ tNavigator	☐ eclipse 300	☐ MORE Roxar
☐ eclipse 100	✓ CMG imex	✓ CMG stars

The keyword sets the initial water saturation. The same number of values as the number of grid blocks should be specified. The values of water saturation should be in the range from 0 to 1.

The keyword has an Eclipse compatible analogue SWAT (see 6.9.9).

Example
```
SW CON 0.3
```

In this example the initial water saturation in all grid blocks is equal to 0.3 (CON (see 7.1.3) specifies the constant value array).

8.6.10 PRES

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	✓	**CMG stars**

The keyword specifies the initial reservoir pressure (*kPa*) for each grid block. The same number of values as the number of grid blocks should be specified.

The keyword has an Eclipse compatible analogue PRESSURE (see 6.9.7).

Example
```
PRES CON 200
```

In this example the same initial pressure is specified in all grid blocks (CON (see 7.1.3) specifies the constant value array).

8.6.11   TEMP

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	√	**CMG imex**	√	**CMG stars**

The keyword specifies the initial reservoir temperature (C) for each grid block. The same number of values as the number of grid blocks should be specified.

The keyword has an Eclipse compatible analogue TEMPI (see 6.9.17).

Example
```
TEMP CON 90
```

In this example the same initial temperature is specified in all grid blocks (CON (see 7.1.3) specifies the constant value array).

8.6.12 CONC_SLD

✓	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	✓	**CMG imex**	✓	**CMG stars**

The keyword specifies initial mole fraction of the components in the solid phase ($gmol/m^3$). One should specify the component name and the value of initial mole fraction of this component in the solid phase for each grid block.

Default: 0.

Example
```
CONC_SLD 'Coke' CON 0.488
```

In this example the same initial mole fraction of the component 'Coke' in the solid phase is specified in all grid blocks (CON (see 7.1.3) specifies the constant value array).

8.6.13   MFRAC_OIL

√	**tNavigator**	☐	**eclipse 300**	☐	**MORE Roxar**
☐	**eclipse 100**	√	**CMG imex**	√	**CMG stars**

The keyword specifies initial mole fraction of the components in the oil phase. One should specify the component name and the value of initial mole fraction of this component in the oil phase for each grid block.

The keyword has an Eclipse compatible analogue XMF (see 6.9.14).

Default: 0.

Example
```
MFRAC_OIL 'DeadOil' CON 0.48223
```

In this example the same initial mole fraction of the component 'DeadOil' in the oil phase is specified in all grid blocks (CON (see 7.1.3) specifies the constant value array).

8.6.14   MFRAC_GAS

☑ **tNavigator**	☐ **eclipse 300**	☐ **MORE Roxar**
☐ **eclipse 100**	☑ **CMG imex**	☑ **CMG stars**

The keyword specifies initial mole fraction of the components in the gas phase. One should specify the component name and the value of initial mole fraction of this component in the gas phase for each grid block.

The keyword has an Eclipse compatible analogue YMF (see 6.9.15).

Default: 0.

Example
```
MFRAC_GAS 'comp4' CON 0.0087
```

In this example the same initial mole fraction of the component 'comp4' in the gas phase is specified in all grid blocks (CON (see 7.1.3) specifies the constant value array).

8.7 Numerical methods control

8.8   Geomechanical Model

8.9 Well and recurrent data

9 Keywords compatible with tNavigator and MORE Roxar

For best adaptation of user experience the keyword notations are chosen to be close as much as possible to the most common simulators:

- Eclipse (c) Schlumberger,

- IMEX (c) Computer Modelling Group Ltd,

- STARS (c) Computer Modelling Group Ltd,

- MORE (c) Roxar.

Eclipse compatible keywords are red. For example: TABDIMS (see 6.1.13). Index of Eclipse compatible keywords – 11.

CMG compatible keywords are pink. For example: TEMR (see 8.4.4). Index of CMG compatible keywords – 12.

MORE compatible keywords are green. For example: IDATe (see 9.1.5). Index of MORE compatible keywords – 14.

The keywords which RFD uses only in tNavigator are blue. For example: REACCONC (see 6.8.46). Index of RFD compatible keywords – 13.

In the description of each keyword there is a table where a simulator is ticked off if this keyword is compatible with this simulator.

tNavigator reads keyword notations of these simulators and converts them into its inner data notations.
This section describes all keywords which can be used in tNavigator:

- tNavigator keywords;

- MORE Roxar keywords;

This description pointed out if there are parameters of the keyword which are ignored by tNavigator or which use is different from CMG.

Data tables are terminated by / symbol. For equal values the standard form NUM*VAL can be used, where NUM – is the number of equal values, and VAL – the value itself.
For convenience keyword are grouped in several sections similar to MORE Roxar sections.

- INPUt Data Section (9.1)

- FLUId Data Section (9.2)

- RELAtive Permeability Data Section (9.3)

- GRID Data Section (9.4)

- INIT Data Section (9.5)

- RECUrrent Data Section (9.6)

MORE Roxar keywords have 4 significant caracters. For convenience (analogous to MORE Roxar) keywords are written in upper case. Sometimes for clarity all caracters of the keyword may be written. For example: FLUI and FLUId. FLUI - the keyword with 4 significant caracters.

If keyword value is not specified by user the simulator uses a default value if it is defined or an error message is issued.

9.1 INPUt Data Section

9.1.1 INPUt

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword begins INPUt Data Section.

9.1.2 TITLe

✓ tNavigator	☐ eclipse 300	✓ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

This keyword specifies a header of the output file. Two title lines may be specified.

Example
```
TITLE First Test run 1971
TITLE Special run
```

9.1.3 PRINt

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword defines printing options for this section.
The following parameters may be specified:

- NONE – no printing of INPUt data,

- ALL – printing of INPUt data.

Example
```
PRIN ALL
```

9.1.4   UNIT

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword specifies the unit system of the simulation.
The following systems are supported:

- METR – Metric units,

- POFU – practical oil field units,

- FIELD – analogue of POFU,

- IMP – analogue of POFU.

The table of units system is in the section 4.
The keyword has tNavigator compatible analogue UNITS (see 6.1.10).

Example
```
UNIT METR
```

9.1.5 IDATe

	tNavigator		eclipse 300	✓	MORE Roxar
	eclipse 100		CMG imex		CMG stars

(✓ tNavigator, ✓ MORE Roxar)

The keyword specifies the initial date of the simulation.

The following parameters should be specified:

1. day of the month (1 or 2 digits);

2. month (first 3 letters of month) – JAN, FEB, MAR, APR, MAY, JUN, JUL, AUG, SEP, OCT, NOV, DEC. JLY (July) is also possible;

3. year (2 or 4 digits); if only 2 digits are specified, a 20th century data is assumed.

The keyword has an Eclipse compatible analogue START (see 6.1.7).

Example
```
IDATE 18 MAY 2003
```

9.1.6   SDATe

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies starting date or time. If the starting time is greater than zero or the starting date is later then the initial date, the run will start at the date SDATe. If the keyword isn't specified, the run will start at the initial date IDATe (see 9.1.5).

The date may be entered in the following formates:

1. value DAYS, where value - the number of days after the initial date IDATe (see 9.1.5);

2. value MONT, where value - the number of month after the initial date IDATe (see 9.1.5);

3. value YEAR, where value - the number of years after the initial date IDATe (see 9.1.5);

4. data format is similar to IDATe (see 9.1.5).

Example
```
SDATE 0 YEARS
```

9.1.7 CNAMe

☑	tNavigator	☐	eclipse 300	☑	MORE Roxar
☐	eclipse 100	☐	CMG imex	☐	CMG stars

The keyword specifies component names in the simulation.

The names of all components should be specified. For black-oil the following names are possible:

- OIL;

- WATer;

- GAS;

- SOLVent;

- STEAM;

The keyword has an Eclipse compatible analogue CNAMES (see 6.7.4).

Example
CNAMe OIL GAS WAT

9.1.8 IMPLicit

☑	**tNavigator**	☐	**eclipse 300**	☑	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword controls the degree of implicitness.

One of the following parameters should be specified after this keyword:

- FULL - fully implicit method;

- ADAP - adaptive implicit method;

- IMPE - IMPES method.

The keyword has an Eclipse compatible analogue IMPLICIT (see 6.1.36).

Example
```
IMPL FULL
```

9.2 FLUId Data Section

9.2.1 FLUId

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

This keyword begins FLUId Data Section.

The type of model should be specified:

- BLACk oil - black oil model;

- EOS - compositional model.

Example
FLUID BLACK

9.2.2   WATR

☑	**tNavigator**	☐	**eclipse 300**	☑	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword sets water properties.

The following parameters should be specified:

1. water density at standard conditions;

2. water density at reservoir temperature and reference pressure;

3. water compressibility;

4. reference pressure;

5. water viscosity at reservoir conditions.

The keyword has Eclipse compatible analogues DENSITY (see 6.5.15), PVTW (see 6.5.4).

Example
```
WATR 999.551 1008.59 4.77175e-05 276.804 0.31
```

9.2.3　BASIc

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify basic fluid properties.

The following parameters should be specified:

1. oil density at stock tank conditions;

2. oil molecular weight;

3. gas molecular weight (if a value greater than 2 is supplied) or gas gravity (If a value less than 2 is supplied).

The keyword has an Eclipse compatible analogue DENSITY (see 6.5.15).

Example
BASI 786.684 190.0 0.792

In this example oil density at stock tank conditions is 786.684, oil molecular weight – 190.0, gas gravity – 0.792.

9.2.4   TEMPerature

✓	**tNavigator**	☐	**eclipse 300**	✓	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword is used to specify temperature for which PVT tables are entered (black oil model).

Example
```
TEMP 160
```

In this example temperature is 160 $^\circ$C.

9.2.5 OPVT

√	**tNavigator**	☐	**eclipse 300**	√	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies oil PVT tables (black oil model). Each table's row should be terminated with a slash /.

Each row consists of the following parameters:

1. oil phase pressure;

2. oil phase formation volume factor;

3. oil phase viscosity;

4. solution gas oil ratio;

5. oil phase compressibility;

6. normalized viscosity slope;

7. surface tension.

The keyword has Eclipse compatible analogues PVCO (see 6.5.5), PVCDO (see 6.5.2), DISGAS (see 6.1.31).

```
Example
OPVT 1.01 1.042 1.040 0.000178 /
17.25 1.130 0.975 0.016119 /
33.48 1.197 0.910 0.032059 /
68.96 1.265 0.830 0.066078 /
137.90 1.425 0.695 0.113277 /
171.38 1.480 0.641 0.138034 /
205.85 1.545 0.594 0.165640 /
273.80 1.675 0.510 0.226197 1.98702e-04 1.30534e-03 /
344.75 1.817 0.449 0.288179 /
620.54 2.337 0.203 0.531474 /
/
```

In this example oil PVT table is specified for 10 pressure values.

9.2.6 GPVT

	tNavigator		eclipse 300	✓	MORE Roxar
✓					
	eclipse 100		CMG imex		CMG stars

The keyword specifies gas PVT tables (black oil model). Each table's row should be terminated with a slash /.

Each row consists of the following parameters:

1. gas phase pressure;

2. gas phase formation volume factor;

3. gas phase viscosity;

4. vapour oil-gas ratio.

The keyword has Eclipse compatible analogues PVDG (see 6.5.6), PVTG (see 6.5.7), VAPOIL (see 6.1.30).

```
Example
GPVT 1.014 935.9505 0.0080 /
18.250 67.8971 0.0096 /
35.487 35.2259 0.0112 /
69.961 17.9498 0.0140 /
138.909 9.0619 0.0189 /
173.382 7.2653 0.0208 /
207.856 6.0637 0.0228 /
276.804 4.5534 0.0268 /
345.751 3.6439 0.0309 /
621.541 2.1672 0.0470 /
/
```

In this example gas PVT table is specified for 10 pressure values.

9.3 RELAtive Permeability Data Section

9.3.1　RELA

✓ tNavigator	☐ eclipse 300	✓ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword begins RELAtive Permeability Section.

Example
```
RELA
```

9.3.2   WETT

☑ **tNavigator**	☐ **eclipse 300**	☑ **MORE Roxar**
☐ **eclipse 100**	☐ **CMG imex**	☐ **CMG stars**

The keyword specifies wettability options and the method of three phase relative permeability calculations.

The following options can be specified:

1. wettability (in order of decreasing wettability): OIL (oil, water and gas) or WATEr (water, oil and gas);

2. method of three phase relative permeability calculations: STN1 (Stone 1 model) or STN2 (Stone 2 model) or LINE (Three phase relative permeability data obtained as straight line interpolation between the two phase relative permeability tables KRWO (see 9.3.3), KRGO (see 9.3.4)).

STN1 is analogous to an Eclipse compatible keyword STONE1 (see 6.6.9); STN2 is analogous to an Eclipse compatible keyword STONE2 (see 6.6.10); LINE is analogous to Eclipse and tNavigator default interpolation of three phase relative permeability data.

Example
```
WETT LINE
```

9.3.3 KRWO

√	**tNavigator**	☐	**eclipse 300**	√	**MORE Roxar**
☐	**eclipse 100**	☐	**CMG imex**	☐	**CMG stars**

The keyword specifies relative permeability tables for water-oil systems. Each table's row should be terminated with a slash /.

Each row consists of the following parameters:

1. water saturation;

2. water permeability;

3. oil permeability;

4. oil-water capillary pressure;

5. ignored, this is a MORE Roxar compatibility field;

6. ignored, this is a MORE Roxar compatibility field.

The keyword has an Eclipse compatible analogue SWOF (see 6.6.1) (four parameters of this keyword correspond to SWOF four parameters).

```
Example
KRWO
0.1200 0.00 1.0 /
0.200 0.00 1.0 /
0.6000 0.3 0.3 /
1.000 1.00 0.0 /
/
```

In this example relative permeability table for water-oil systems is specified for 4 pressure values.

9.3.4   KRGO

✓ tNavigator	☐ eclipse 300	✓ MORE Roxar
☐ eclipse 100	☐ CMG imex	☐ CMG stars

The keyword specifies relative permeability tables for gas-oil systems. Each table's row should be terminated with a slash /.

Each row consists of the following parameters:

1. gas saturation;

2. gas permeability;

3. oil permeability;

4. gas-oil capillary pressure;

5. ignored, this is a MORE Roxar compatibility field;

6. ignored, this is a MORE Roxar compatibility field.

The keyword has an Eclipse compatible analogue SGOF (see 6.6.2) (four parameters of this keyword correspond to SGOF four parameters).

```
Example
KRGO
0.0000 0.0000 1.0000 /
0.0200 0.0000 0.9970 /
0.0500 0.0050 0.9800 /
0.1200 0.0250 0.7000 /
0.25 0.1250 0.2000 /
0.3 0.1900 0.090 /
0.45 0.6000 0.0100 /
0.5 0.7200 0.0010 /
0.7 0.9400 0.000 /
0.88 1.0000 0.0000 /
/
```

In this example relative permeability table for gas-oil systems is specified for 10 pressure values.

9.4   GRID Data Section

9.5 INIT Data Section

9.6   RECUrrent Data Section

10　Data files

The program may be started either directly, or from command line with one argument – name of data file.

- The simulator uses the following files:

  - `<name>.data` — file with initial data, should be passed to the program as an argument,

  - `<name>.inc` — files included into data file (if they exist).

- The simulator tNavigator 3.2 generates the following files:

  - `<name>.log` — text file with report on simulation process;

  - `<name>.err` — text file with report on all errors in input data and during simulation;

  - `<name>.rst, <name>_*.res,` — binary files with simulation results (pressure and saturation maps, well modes description, etc.); these files are also used restarting previously carried out computations (RESTART (see 6.1.8)).

10.1　Data file

Initial data file contains description of input parameters. It is written on keyword language (see 6).

10.2　Files with simulation results

Simulation result binary files produced by tNavigator 3.2 may be viewed using tNavigator post-processing environment.

Text files created by simulator may be viewed using any program that can work with text files.

11 Keyword definitions index Eclipse

12 Keyword definitions index CMG

13   Keyword definitions index RFD

14　Keyword definitions index MORE

Список литературы

[1] Aziz, K., Settari A., Petroleum reservoir simulation, *Applied Science Publishers LTD*, London, 1979.

[2] Ertekin, T., Abou-Kassem, J.H., King, G.R., Basic applied reservoir simulation. *SPE Textbook Series*, Richardson, Texas, 2001.

[3] Stone, H. L., Probability Model for Estimating Three-Phase Relative Permeability. Trans AIME (JPT), 249, p. 214-218, 1970.

[4] Stone, H. L., Estimation of Three-Phase Relative Permeability and Residual Oil Data. Can.Pet.Tech., Vol 12, p. 53-61, 1973.

[5] Carter, R. D., Tracy, G. W., An Improved Method for Calculating Water Influx. Trans AIME (JPT), 219, p. 58-60, 1960.

[6] Fetkovich, M. J., A Simplified Approach to Water Influx Calculations - Finite Aquifer Systems. JPT, p. 814-828, July 1971.